

École polytechnique de Louvain

Numerical modelling of wood-wood assemblies for multi-storey buildings

Author: Tristan BAUDOUX

Supervisor: Pierre Latteur

Readers: Benjamin Dardenne, Justin Goosse, LUCA Sgambi, Serena Van Nimwegen

Academic year 2024–2025

Master [120] in Civil Engineering

Page left blank for printing purposes.

Abstract

Multi-storey buildings are indispensable in modern life, serving as residences, offices, and educational institutions. However, the construction sector is a significant contributor to climate change. Wood, with its exceptional environmental properties, offers a sustainable alternative to conventional materials like concrete and steel. Yet, its widespread adoption is hindered by higher costs. A potential solution lies in minimizing processing by using locally machined debarked circular logs. This approach presents another challenge: the complexity of designing wood-wood assemblies, which often necessitates costly experimental campaigns. This master's thesis explores the development of simplified numerical models for wood-wood assemblies to predict load transfer behaviour. While achieving a fully accurate numerical representation of wood requires further research, this work contributes progress by leveraging the commercial finite element software Abaqus. Elastic properties and orthotropic behaviour of wood are modelled, and material model comparisons are conducted to help users navigate the software. Key focus areas include the study of contact rules, tolerance challenges, and simplified modelling of wood's non-linearities. The methodology is designed to be accessible for engineers and researchers with limited numerical modelling expertise. The final numerical model is tested using a pull-out test on a complex wood-wood assembly, with experimental results for comparison. Despite its simplicity, the model captures certain fracture modes and approximates the progressive loss of strength and final failure reasonably well. Limitations include the inability to accurately model crack behaviour under tension and the need for calibrated elastic and plastic parameters specific to the tested wood. Future improvements could enhance the model's accuracy and applicability in practical settings.

Key words: wood, circular timber, logs, wood-wood assemblies, numerical modelling, Abaqus, orthotropic behaviour, non-linear analysis, experimental comparison

Page left blank for printing purposes.

Acknowledgements

Before delving into the subject of this master's thesis, I would like to express my heartfelt gratitude to the individuals and institutions whose support has made this work—and the completion of my studies—possible.

First and foremost, I extend my deepest thanks to my supervisor, Professor Pierre Latteur, for granting me the opportunity to explore this fascinating topic with remarkable freedom. I am sincerely grateful for your invaluable guidance, insightful advice, and innovative ideas throughout the development of this thesis. Your ability to spark my interest in civil engineering at the outset of my studies and later inspire my passion for the remarkable material that is wood has been truly transformative.

I am also highly grateful to Justin Goosse, whose dedication has been nothing short of extraordinary. Your unwavering patience, constant availability and thoughtful feedback have been a true source of motivation. Your generosity in sharing your expertise and your encouragement throughout this journey have been invaluable.

I also wish to thank Serena Van Nimwegen for her constructive feedback and innovative ideas, always offered with a willingness to help and a collaborative spirit that made our exchanges a pleasure. My gratitude extends to Martin Steinmetz as well, for his expert advice and assistance with the complex functionalities of Abaqus, which were instrumental to this work. This thesis would also not have been possible without the resources and facilities provided by my university, Université catholique de Louvain (UCLouvain), for which I am deeply thankful.

On a personal note, I am profoundly grateful for the steadfast support of my family and friends throughout this journey and my entire academic career. My deepest thanks go to my parents, Véronique and Denis, whose unwavering guidance, optimism, and belief in me have been the foundation of my success. You have afforded me the privilege of pursuing a fascinating university education, and for that, I will forever be thankful.

To Aleksandra, your unfailing support, patience, and faith in me have been a source of strength and comfort. I am deeply appreciative of everything you have done to encourage and motivate me.

Finally, I would like to thank the members of the jury for taking the time to review this thesis and for the valuable feedback you will provide.

Page left blank for printing purposes

Table of Contents

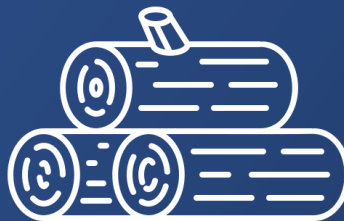
1. INTRODUCTION – LOGS AS A BUILDING MATERIAL	9
1.1. ROBOTRONC PROJECT	10
1.1.1. CONTEXT OF THE PROJECT (1)	10
1.1.2. THIS THESIS IN THE ROBOTRONC PROJECT	12
1.1.3. EXISTING WORK WITHIN THE ROBOTRONC PROJECT	12
1.2. THE IMPORTANCE OF THE ASSEMBLIES	13
1.3. ENVIRONMENTAL CONSIDERATIONS	15
1.4. FOREST PRODUCTION AND MANAGEMENT	19
1.5. THE COMPLEX PROPERTIES OF WOOD	26
2. NUMERICAL SIMULATIONS OF WOOD	35
2.1. A SHORT STATE OF THE ART OF NUMERICAL ANALYSIS OF WOOD	37
2.2. INTRODUCTION TO THE FINITE ELEMENT METHOD AND ABAQUS	47
2.2.1. THE FINITE ELEMENT METHOD	47
2.2.2. THE COMMERCIAL SOFTWARE ABAQUS	52
2.3. MATERIAL MODELS	53
2.4. INTRODUCTORY EXERCISES OF WOODEN BEAMS	56
2.4.1. EXERCISE ONE: SIMPLY SUPPORTED BEAM	56
2.4.2. EXERCISE TWO: OVERHANGING BEAM	64
2.5. KEY PARAMETERS OF NUMERICAL SIMULATIONS OF WOOD	69
2.5.1. CONTACTS	70
2.5.2. TOLERANCES	78
2.5.3. NON-LINEARITIES	81
3. ROBOTRONC’S WOOD-WOOD ASSEMBLY	91
3.1. EXPERIMENTAL PULL-OUT TEST CONDUCTED IN JULY 2024	91
3.1.1. OVERVIEW OF THE ASSEMBLY AND TESTING PROCEDURE	91
3.1.2. EXPERIMENTAL RESULTS ANALYSIS AND INTERPRETATION	101
3.2. NUMERICAL SIMULATION OF THE PULL-OUT TEST	104
3.2.1. NUMERICAL TEST PARAMETERS AND CONFIGURATION	104
3.2.2. NUMERICAL RESULTS ANALYSIS AND INTERPRETATION	108
3.3. COMPARISONS BETWEEN THE EXPERIMENTAL AND NUMERICAL RESULTS	113
4. CONCLUSION	119

5.	<u>ANNEXES</u>	<u>123</u>
5.1.	MOST COMMON FAILURE CRITERIA	123
5.2.	DETERMINATION OF THE ORTHOTROPIC PARAMETERS	126
5.3.	MESH CHOICE AND MESH CONVERGENCE STUDY	126
5.4.	TUTORIAL – KEY STEPS IN ABAQUS CAE	130
6.	<u>LIST OF ABBREVIATIONS</u>	<u>134</u>
7.	<u>LIST OF FIGURES</u>	<u>135</u>
8.	<u>LIST OF TABLES</u>	<u>140</u>
9.	<u>BIBLIOGRAPHY</u>	<u>142</u>

Chapter 1

Introduction

Logs as a building material



1.Introduction – Logs as a building material

Construction plays a fundamental role in shaping our society, providing the infrastructure and spaces where we live, work, and play. From skyscrapers to homes, construction projects are the backbone of modern civilization, driving economic growth, innovation, and urban development. Since the beginning of human civilization, buildings have offered us a shelter.

However, the construction industry also carries a significant environmental footprint. Traditional construction methods often rely heavily on resource-intensive materials like concrete and steel, contributing to carbon emissions, habitat destruction, and depletion of natural resources. Additionally, construction waste accounts for a substantial portion of global landfill content, exacerbating environmental challenges.

One promising solution to mitigate the environmental impact of construction is embracing wood as a primary building material. Wood offers a renewable and sustainable alternative to traditional materials, as trees can be replanted and harvested in a cyclical manner. Furthermore, using wood in construction can sequester carbon, helping to combat climate change by storing atmospheric carbon dioxide within the built environment.

Building with wood also offers numerous benefits beyond its environmental advantages. Wood is lightweight, versatile, and aesthetically pleasing, allowing for innovative architectural designs and efficient construction processes. Additionally, timber construction techniques, such as mass timber and engineered wood products, enable the creation of high-performance buildings that meet or exceed the standards of traditional construction methods. However, this increase in performance through transformations comes with an increase in cost as well as polluting emissions. A more sustainable alternative is to transform the wood the least possible by using debarked logs.

Embracing wood as a primary building material represents a holistic approach to sustainable development, balancing the needs of society, the economy, and the environment. By promoting responsible forestry practices and advancing wood-based construction technologies, we can build a more resilient and harmonious future for generations to come.

This master's thesis is part of the project “RoboTronc”¹ which seeks at building multi-storey buildings using wooden logs milled by robotic arms. More specifically, this thesis focuses on the column-column-beams assemblies required to build these buildings. The objective is to determine the best assembly for the structure by using only wood if possible. The interest of building using minimally processed wood in contrast to squared timber or

¹ “RoboTronc” is the combination of the word “Robot” and the word “Tronc” meaning trunk in French.

engineered timber is to reduce the cost² as much as possible in order to make timber highly competitive and become a reference material for buildings. By removing the energy-intensive transformations from the process, the environmental impact of the wood is highly reduced resulting in more sustainable structures.

In truth, building using wood is a challenging process for now for multiple reasons: inexperience of the structural engineers, architects and construction companies, higher costs, etc. However, after gaining more experience through research and exemplary projects as well as significantly reducing the costs by using new techniques like minimum transformations of the wood as well as robotic arms to shape the assemblies. This way, timber construction can become a more economical, practical and generally a more compelling option than its usual competitors.

1.1. RoboTronc project

1.1.1. Context of the project (1)

The RoboTronc project (1) consists of a collaborative project between the UCLouvain, MOBIC³, DUCHÊNE⁴ and the Buildwise⁵ with the help of the SPW – Service Public de Wallonie. The main goal of the project is to find technical solutions to build multi-storey buildings using logs in order to make it the most environmentally conscious way of building as well as the most cost-effective making it the clear choice for future builders. (Figure 1)

Currently, most timber buildings are built using either squared timber or engineered timber as it is easier to joint, standard in dimensions and it is part of the norms like the Eurocode 5 (EC5). (2) However it has mostly three disadvantages. Firstly, a significant part of the wood (ranging from 30 to 60[%] depending on several factors) is either wasted or used for other tasks like burning or panel making. Secondly, the wood which has been squared has a reduced section compared to the original log and a part of the fibres constituting the wood are cut in the process, making it less strong than originally. Finally, squared timber and engineered timber require more energy than a simple debarked log making it more costly and more polluting. The transformations required for squared timber or engineered timber currently make wood more expensive than its equivalent in concrete or steel which explains their prevalent use.

² Highly transformed wood can cost more than 1000 euros/m³.

³ Company specialized in prefabrication and construction of wooden houses and buildings by using robotic arms.

⁴ General construction company working on multiple types of projects: buildings (publics, private, residential, offices, industrial, etc.) as well as civil construction works of major size, carpentry works and rail electrification.

⁵ Company supporting the construction industry by developing scientific research, making it available publicly, bring a technical support to companies and members as well as conduct research upon request of the public sector.

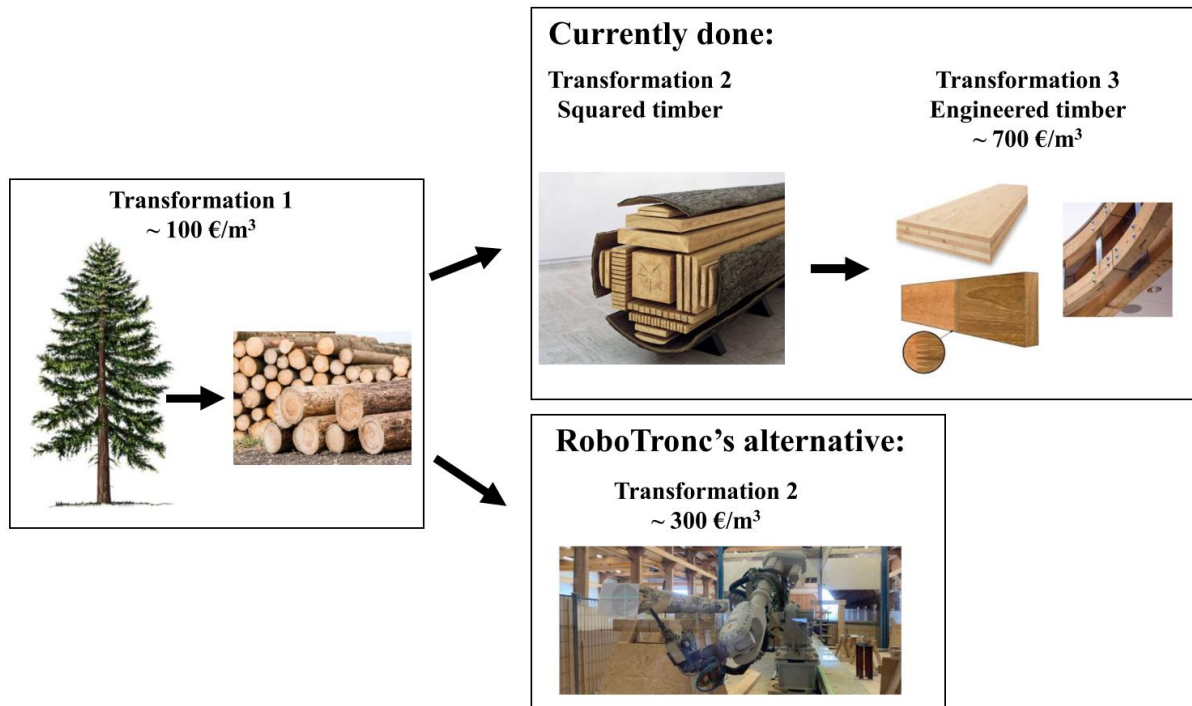


Figure 1: Current way of building vs. the RoboTronc's alternative. (1) (3)

In order to build using logs, multiple challenges need to be overcome:

- The first challenge is to determine the desired level of log drying. Complete drying minimizes the risk of fungal and insect infestations while reducing the impact of volumetric changes on assemblies. However, achieving this requires significant time or energy. Alternatively, a lower level of drying is under review.
- The logs can be individually tested through a new process in order to know their real individual resistance. This new non-destructive process needs to be determined.
- The logs can be scanned⁶ in 3D in order to know their exact geometry and optimize their use.
- Once their geometry is known, a robotic arm⁷ will be used to carve the wood and make the joints.

It is important to note that while the project advocates for wood utilization, it does not seek to eliminate the use of concrete or steel in some parts of the construction but to reduce it. Concrete and steel will certainly still be necessary for critical elements like foundations,

⁶ This step is optional and still under review.

⁷ The robotic arm in question is a 6-axis model, commonly used in the automotive industry. It can be equipped with a variety of milling tools at its end effector. The arm's movements and operations are controlled by a computer program. See (3) for more details on the functioning of this device.

The electric consumption of such robotic arms in order to mill the assemblies still requires to be studied.

underground car parks, central cores or screeds allowing for an efficient diaphragm effect and the transfer of the loads.

1.1.2. This thesis in the RoboTronc project

This master's thesis is only a very small part of the research required to achieve this result. It is organized into three main chapters and a conclusion, each addressing a key aspect of the research in the field of wood-wood assemblies.

Chapter One The first chapter serves as a general introduction by providing an overview of the RoboTronc project and the importance of the assemblies for timber buildings. The chapter also explores the environmental considerations linked to building using wood as well as the implications on forests and their management. The chapter concludes with a discussion of the complex properties of wood which are critical to know when designing and building timber buildings.

Chapter Two The second chapter delves into numerical simulations of wood beginning with a short State of the art outlining the latest advancements in the field. An introduction to the Finite Element Method (FEM) is also presented as this computational technique forms the basis of the numerical analyses here conducted using the commercial software Abaqus 2021. Afterwards, the material models considered in the FEM software are detailed and compared through a series of introductory exercises. Lastly, three subsections address key parameters in Abaqus: contact between elements, the impact of tolerances and the modelling of the non-linearities of wood.

Chapter Three The third chapter focuses on the numerical modelling of a so called “RoboTronc's assembly” corresponding to complex geometries of wooden beam and column. First, an experimental pull-out test made by the RoboTronc team and its results is described followed by its numerical reproduction. Finally, the two situations are compared to analyse the strengths and weaknesses of the current numerical model of wood developed. The objective is qualitative, with the focus being on understanding the assembly's behaviour rather than precisely determining its strength. Additionally, the numerical model configured is designed to be reproducible.

Finally, the chapter four provides a summary of the work carried out in this master's thesis, synthesizing the key findings and highlighting potential avenues for further research to refine and enhance the modelling of the material wood.

1.1.3. Existing work within the RoboTronc project

Many students and researchers from the UCLouvain have worked on the RoboTronc project and the interested readers of the present master's thesis are invited to take a look at all

the interesting work achieved till now. The following references are placed in a chronological order.

- Goosse, 2020 (3): Experimental validation of assemblies of reciprocal structures of whole logs.
- Moens and Niclaes, 2021 (4): Mechanical characterisation of logs.
- Geno, Goosse, Van Nimwegen, and Latteur, 2022 (5): Parametric design and robotic fabrication of whole timber reciprocal structures.
- Dehaen and Ponthière, 2022 (6): Search of a non-destructive experimental protocol of mechanical characterisation of logs.
- Houben, 2022 (7): Mechanical non-destructive characterisation of logs using the GrindoSonic.
- Dupriez and Stevens, 2022 (8): Numerical modelling of whole log wooden floors
- De Coster, 2023 (9): Recherche d'assemblages colonne-colonne-poutres pour bâtiments multi-étagés en bois rond.

1.2. The importance of the assemblies

Key aspects of the section: Assemblies are crucial to structural safety, cost, and durability, particularly in wood structures, where material complexities like anisotropy, hygroscopicity, and fire sensitivity demand careful design. A statistical research attributes 25% of timber structure failures to assemblies, underscoring the need for improved engineering, construction practices, and education. Key recommendations include enhancing design understanding, controlling timber production quality, and developing robust, resilient structures to mitigate human errors and local failures.

Whatever the material used to build a structure, the assemblies are always a key element for many reasons: safety, cost, sustainability of the structure, etc. Therefore, they must be treated with utmost care. This is even more important for structures principally made out of wood due to the high complexity of the material: sensitivity to hygrometry, insects and mushrooms, changes of volume and strength dependent to the direction of the fibres, etc.

The following list identifies some of the characteristics which an assembly should respect. The assembly should be:

- **simple.**
In truth, it must be understood by the designers as well as the workers. The transfer of forces through the structure must be clear.

- **able to handle all the steps of the construction.**
Meaning that it will not be damaged or fail at an intermediate step of the construction.
- **easy to install.**
Thus reducing risks of bad assembly, higher costs and time of construction.
- **mostly unaffected by hygrometric and thermal variations.**
By allowing slight movements of the assemblies, the structure has little consequences due to the changes of volume of its elements.
- **resistant to fire, to a certain extent.**
Care must be taken regarding steel parts quickly losing their strength in the event of fire.
- **ductile.**
Ductile assemblies allow predictable failure behaviours allowing users to evacuate or repairs to be done in time.
- **long-lasting.**
A well-designed assembly should maximize the structure's lifespan, helping to save money and reduce the carbon footprint of construction.
- **behave adequately under earthquake, flood or windstorm situations.**
Nowadays, extreme events are more and more frequent and our modern structures must be able to protect its users for decades to come.

E. Frühwald (10) (11) (12) explains in her articles and lectures the importance of studying the previous ruptures in timber structures to assess their weaknesses and improve them. Her work has been partially based on statistical studies of failures concerning timber structures like the two following: (13) and (14). The article Frühwald (12) analyses that around 25% of the failures were due to the assemblies themselves whereas only 4% concerned the columns alone. Moreover, the same article highlights that structural mistakes are mainly due to either a mistake in the design/conception/planification phase or in the construction phase. They observed that the ratio between those 2 phases is around 50%. This highlights the importance of increasing the knowledge of the engineers with regards to the assemblies and their behaviour in the structure.

Frühwald identified the following elements as being critical in timber engineering. The first one concerns wind bracing for which she insists that sequences of construction must be studied as well as clear instructions given to the workers. The second one concerns risky situations for failures perpendicular to the fibres of wood. She suggests that engineers must be more educated regarding the anisotropy of wood and its drying properties. Thirdly, the effects of humidity on wood are also a critical parameter of wood. She suggests that design engineers and control engineers verify the consequences of assemblies which are impeached to grow due to an increase of humidity. The study of hygroscopicity must be preponderant in timber

engineering. Finally, she specifies that the design of the assemblies must be better understood and studied. She suggests that the production of timber elements must be more carefully controlled to reduce the risks of having a material behaving less well as expected.

Additionally, the study explains that none of the failures identified were due to an unfavourable combination of load events. This signifies that the level of safety of the norm is sufficient and that the causes are not due to the norms but other factors. They identify the following needs and solutions: improvement of construction procedures, increase of controls, increase of training of the workers on site as well as increase of the education of the designers. Finally, the author insists that the future research in timber engineering must be linked to studying more robust and resilient structures which are less sensitive to the failure of local elements as well as human mistakes.

1.3. Environmental considerations

Key aspects of the section: Climate change, driven by human activity, is a pressing challenge that calls for urgent action. Construction is a great sector to improve as it accounts for 39% of global carbon emissions. Timber, when sourced sustainably, offers a valuable way to reduce embodied carbon in buildings by storing carbon absorbed during tree growth. With proper design, construction, and maintenance, timber structures can last for centuries, amplifying their environmental benefits.

This chapter aims at answering the following questions: why do we need more environmentally sustainable materials? Is timber one of them and why?

First of all, it is nowadays not a surprise to anyone that climate change is real and is happening even quicker than anticipated. Indubitably, as we can read in the latest report of the GIEC: “Global greenhouse gas emissions have continued to increase, with unequal historical and ongoing contributions arising from unsustainable energy use, land use and land-use change, lifestyles and patterns of consumption and production...”. (15) The causes of the Climate Change are multiple and in order to effectively reduce our impact on Earth, we must act on as many sectors as possible.

One significant sector participating to climate change is construction and buildings in general. As a matter of fact, buildings are currently responsible for 39% of global energy related carbon emissions with 28% being Operational Carbon (OC)⁸ and 11% being Embodied Carbon (EC)⁹ (16). The fact that buildings significantly contribute to climate change informs us on the

⁸ Operational Carbon (OC) refers to the emissions produced during the use of the building: heating and cooling, electricity consumption, ventilation, etc.

⁹ Embodied Carbon (EC) refers to the emissions produced for the entire life cycle of a material: from the extraction of raw material, its installation in the building and finally, its disposal or recycling.

importance of optimizing such sector (see Figure 2). By focusing our energy as engineers, architects, clients and policymakers on improving the carbon emissions of buildings, we can have a strong and quick impact on reducing our emissions.

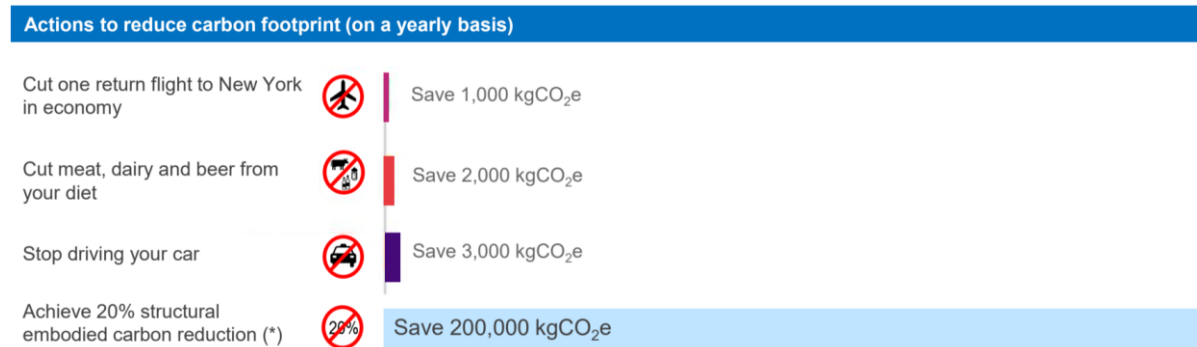


Figure 2: Comparison of actions to reduce our carbon footprint (on a yearly basis). (77)

Furthermore, the urgency of Climate Change is being shown to us by its consequences (see Figure 3 and Figure 4). Once again, we can read in the latest report from the GIEC: “Widespread and rapid changes in the atmosphere, ocean, cryosphere and biosphere have occurred. Human-caused climate change is already affecting many weather and climate extremes in every region across the globe. This has led to widespread adverse impacts and related losses and damages to nature and people (high confidence). Vulnerable communities who have historically contributed the least to current climate change are disproportionately affected (high confidence).” (15). Additionally and linked to the increase of extreme weather events, many ecosystems and species are highly impacted, on land as well as in the ocean. Food and water security is reduced, food-borne and water-borne diseases are increasing as well as mass movements of populations. It is important to note that Climate Change already has a strong impact on our society and that in order to reduce the future consequences, we must act now.

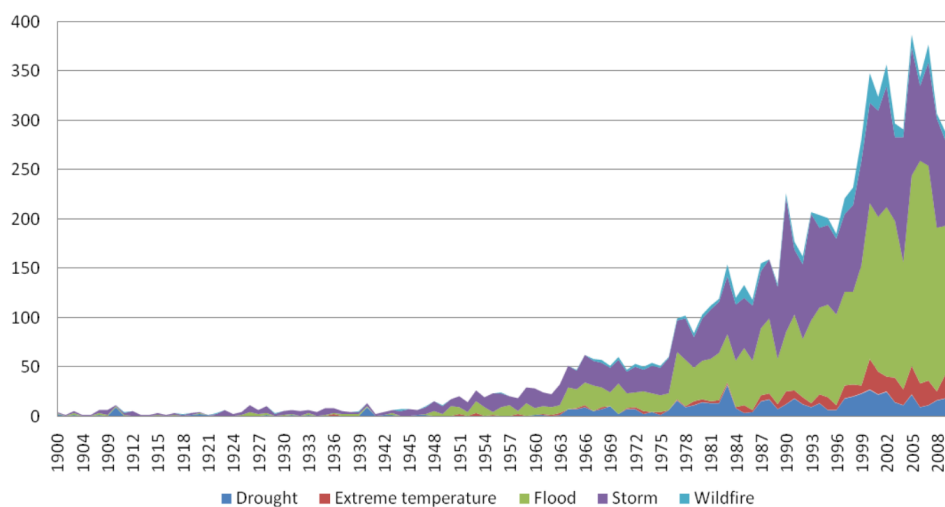


Figure 3: Number of extreme weather events globally, by year. (78)

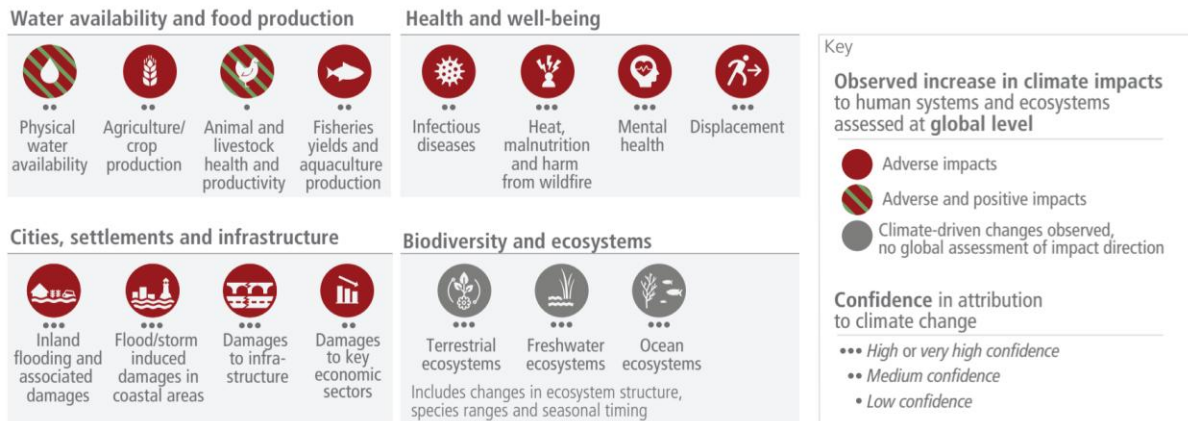


Figure 4: Observed widespread and substantial impacts and related losses and damages attributed to climate change. (15)

Moreover, as visible in the Figure 5, the World's population is likely to increase from 8 billion people to approximately 10 billion. Therefore, not only do the current buildings highly need a proper renovation to face the Climate Change but we will also need to build more places to live and work for the additional 2 billion people who will be on our planet. The relevance of optimizing and improving the way we build is becoming more and more clear.

There exist multiple ways to improve the way we build: energy efficiency (insulation), water consumption, waste management, sustainable urban planning (Mixed-used developments, green spaces, efficient public transport, ...), building automation, etc. One of them, on which this master's thesis focuses, is the use of more sustainable materials to reduce the Embodied Carbon of our buildings. In fact, one material stands out when it comes to sustainability: wood. But why is that?

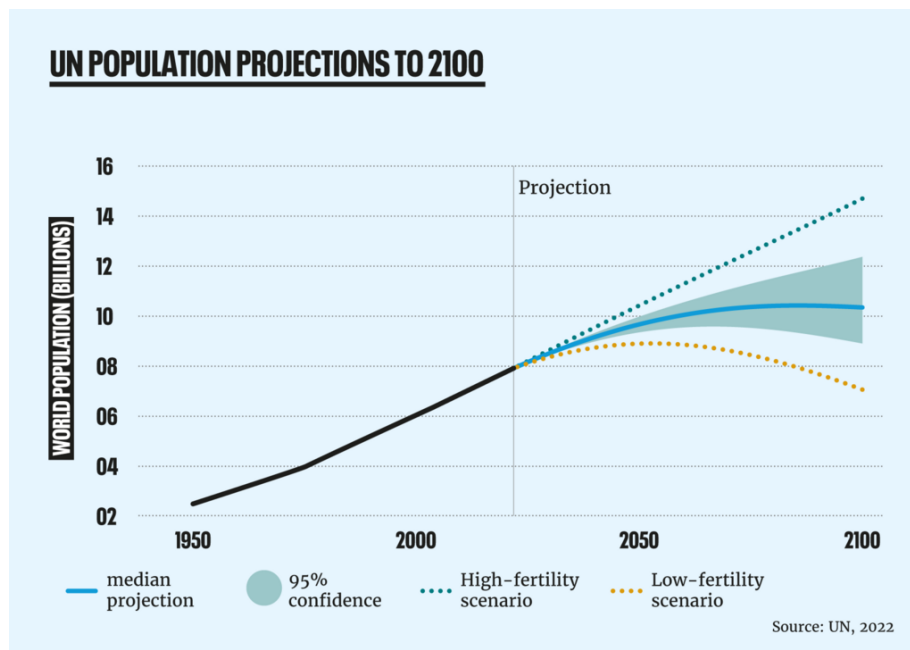


Figure 5: World population (Billions) projections by the UN (2022).

Wood, when the forests are managed in a sustainable way (see following chapter), can significantly help us reduce the carbon footprint of renovated or new buildings. Through photosynthesis, trees store carbon from the CO_2 in the air when growing (see Figure 6) (17). This way, the timber used in construction serves as a useful storage of carbon. At the end of life of the wood, if it is incinerated or decomposed, the CO_2 is resituated to the atmosphere. When unexploited, the wood in forests rots and restores most its carbon to the atmosphere and a small part in the ground through its roots. It is also interesting to note that the rate of carbon sequestration depends on the age and species of the trees. As it happens, young and mature trees have a higher rate of carbon sequestration than old trees.

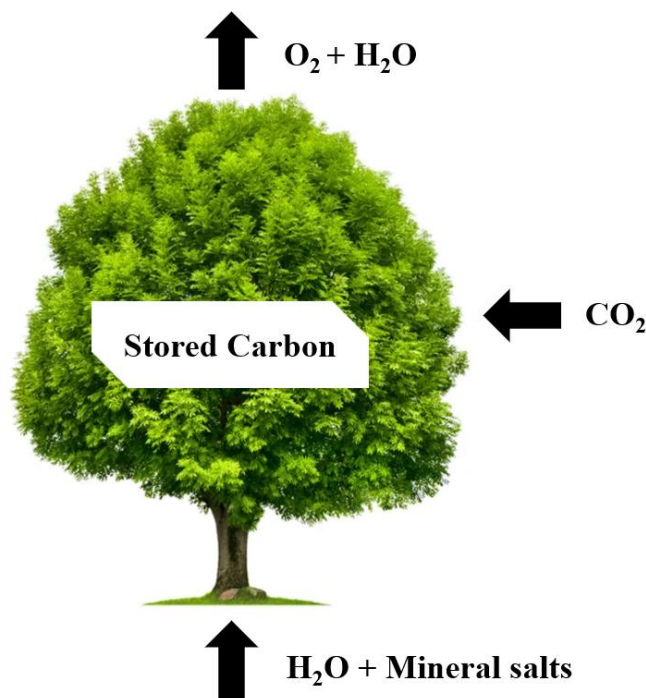


Figure 6: Diagram of the chemical flows created by trees (17).

Therefore, it is important to impeach deforestation and even better: create forests. Trees act as a source of production of O_2 , a permanent storage of CO_2 in the wood, a mean of permanent storage of Carbon in the ground. The use of wood in construction is pertinent because it allows another mean of Carbon storage but it is only conceivable if forests are well managed and replanted intensively (see next chapter regarding Forest production and management) (17) (18). It is also important to note that in order for the timber to be beneficial to the environment, the building must be well designed, built and maintained for it to last as long as possible: 100 years, 200 years or even more. This is possible through a comprehensive study of the material, its assembly in buildings and its maintenance. For example, the species of wood must be compatible with the hygrometric environment of the building to reduce the risk of insect attacks or fungi.

Additionally, wood offers many other advantages making it a competitive material: it is a renewable resource, it is versatile (also through the use of modified wood products like CLT or Glulam), light, strong, fire-resistant and allows for a circular economy (19). Lastly, wood also allow for a significant robot potential allowing an increase in productivity both during the production and installation as well as a high-level of flexibility (3) (5). For a more in-depth study of the environmental benefits of using timber in construction, we invite the reader to the numerous sources mentioned in the bibliography notably regarding the Life Cycle of wood (16), from (19) to (20). Finally, it is important to note that there are scenarios where timber is not pertinent such as for underground car parks where concrete is an excellent material of choice due to its weight, its imperviousness when properly selected, etc. Steel can also still be pertinent for large-scale infrastructures such as long-span bridges or skyscrapers.

1.4. Forest production and management

Key aspects of the section: To transition construction practices toward timber, specific criteria must be met, including sustainable forest management. Forests are crucial for biodiversity, carbon storage, and timber supply, but face threats from agriculture, climate change, and illegal deforestation. A proper forest management and certifications like PEFC ensure long-term viability. Timber's circular potential and minimal processing, as emphasized in RoboTronc, offer a promising future for eco-friendly construction while safeguarding forest ecosystems.

It is important to note that if we want to change our habits of construction (new buildings as well as renovations) towards timber, there needs to be certain criteria met. In fact, a significant reason why timber is a construction material to consider for today and the future is its significantly lower carbon footprint as well as carbon storage. However, this can only be achieved thanks to properly managed forests.

First of all, what is the current status of the forests of the World and in Belgium? In the Figure 7, we can observe the proportion and distribution of the forests around the world. As we can see, Tropical forests are mostly spread around South America and Central Africa whereas Boreal forests are located mostly to the north of North America, North Europe and Russia. Temperate forests are mostly visible in North America and Europe. Subtropical forests are mainly in Eastern Asia.

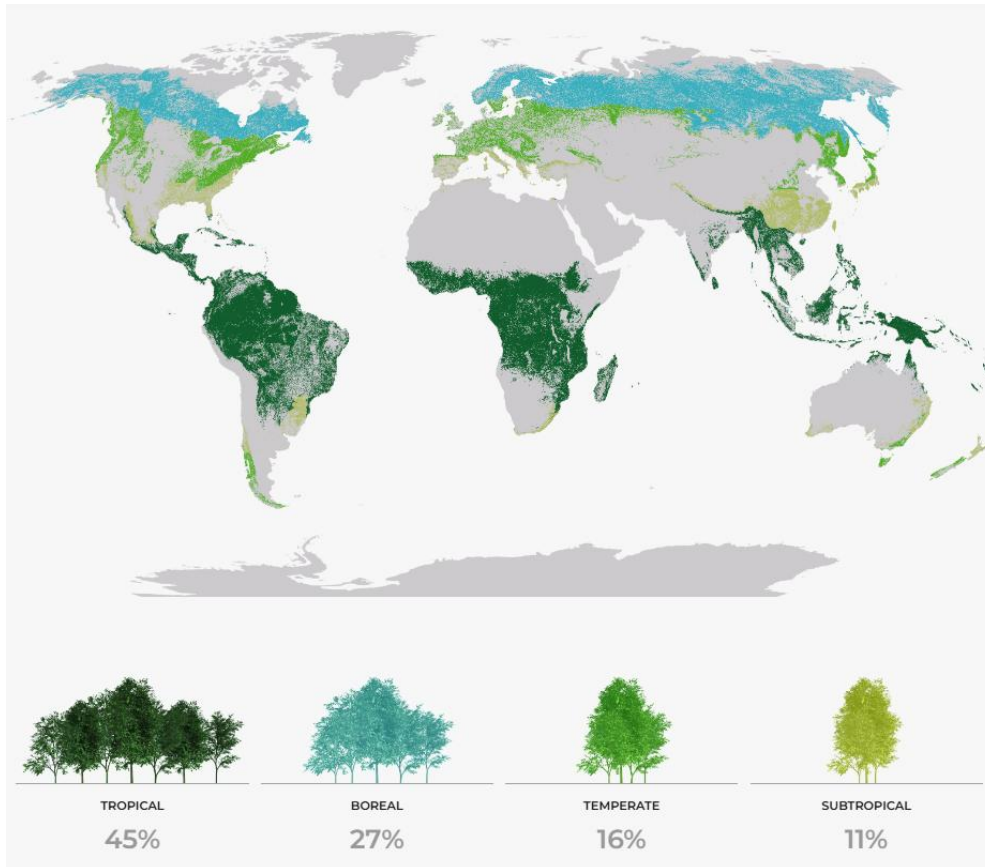


Figure 7: Proportion and distribution of global forest cover by climatic domain in 2020. Source: FAO.

As we can see, forests are not evenly spread on Earth and are therefore a key resource for some countries and their economy. In the Table 1, we can observe the precise forest areas of the different geographical areas, their proportion to the world and the proportion of their surface covered by forests. This highlights that South America has the most significant forest area with 940 million hectares followed by Russia with 815 million hectares together accounting for 43% of the forests of the world. The 5 countries with the biggest forest areas are the following: Russia, Brazil, Canada, USA and China.

It is important to note that forest proportions around the world have an impact on the geopolitical influences and safety. It is interesting to note that Asia is currently using significantly more timber than they are producing and therefore importing it from the other areas of the world. Additionally, in 2018, 4 billion m³ of wood has been harvested. 40% of these were destined for heat or energy production whereas 60% were destined for construction. (21) (22)

	Forest area [10 ⁶ ha]	% of the forests of the world	% of ground covered by forests
Africa	637	16	21
North America	657	16	32
South America	940	23	51
Asia	623	15	20
Europe	202	5	35
Russia	815	20	50
Oceania	185	5	22
World	4 059	100	31

Table 1: Forest areas of the World. Source: FAO, 2021.

Next, the current status of the Belgian forests is described. In Table 2, we can observe the main parameters: Total area of the 3 main regions of Belgium, their forested areas and afforestation rate, the proportion of public/private owned forests as well as the type of wood taking part in these forests. Belgium as a whole has a little less than a quarter of its surface covered by forests (compared to 31% average for the world). However, we can see that most of the forests of Belgium are located in the southern half of the country with 563 thousand hectares of forests against 140 thousand hectares for the northern half of the country.

	Total area [ha]	Forested area [ha]	Afforestation rate [%]	Public forests [ha]	Private forests [ha]	Hardwood [%]	Softwood [%]
Brussels	16 140	2 210	14	95	5	92	8
Flanders	1 352 230	140 279	10	36	64	61	39
Wallonia	1 684 430	563 000	33	49	51	57	43
Belgium	3 052 800	705 489	23	47	53	58	42

Table 2: Forest areas of Belgium. Source: Panorabois, 2021, OEWB.

Additionally, around half of the forests are privately owned. Finally, we can see that Belgium has approximately 60% of Hardwood compared to 40% of Softwood. The most common Hardwood trees in Belgium are Beech, Oak and Poplar whereas the most common Softwood are Spruce, Larch, Scots pine and Douglas. Their ideal exploitation age can range from 25 years (Poplar) to around 100 years (Softwood) or 200 years (Hardwood). In absence of silviculture, these trees can live for centuries up to a thousand years or more.

In Belgium, around 80% of the harvested wood is destined to construction whereas the remaining 20% are aimed to be burnt. Finally, it is interesting to note that the forested area of

Belgium has grown by 59% in 140 years. Moreover, Belgian productivity to harvest wood is 10 times higher than the World average as Belgian forests have around 7m³/ha. (21)

What is the purpose of the forests on Earth? Forests through their trees, produce a significant amount of oxygen and store a lot of carbon dioxide from the air. Another important factor is the biodiversity that they protect. As a matter of fact, they house 80% of the amphibian species, 75% of the bird species and 68% of the mammal species of the world. Forests are also essential to human food security and many communities. (22) Through a shift of materials used in construction, forests will also become an essential asset to harvest the best way possible in a sustainable way to be able to protect the biodiversity it holds and use its materials for centuries to come.

In order for a shift of materials used in construction to happen, timber needs to be competitive and to be able to answer to a growing strong demand. This can only happen through a proper forest management where the replanted species of trees are carefully chosen (diverse species, strong growth, resiliency to drought and insects, etc.), the spacing between the trees is carefully studied as well as thinning techniques allowing to obtain the desired types of trees (long and thin or short and thick) and optimizing the productivity of the forests. For the project RoboTronc, it is important to note that the circular tree trunks planned to be used have to respect stricter criteria of shape and dimensions compared to squared timber or engineered timber. The expectable tree trunks diameters are planned to be ranging from 300 [mm] to 600 [mm] of diameter in order to keep a low weight, to be easily machined at the joints, to keep a low cost, to be easily available and to limit the problems linked to shrinkage. (9) (23)

When discussing about the future of timber in construction, many people are still concerned about deforestation. And this is good because deforestation is a significant threat to our life on Earth and it needs our attention. However, it is worth noting that the harvesting of timber for construction purposes or energy currently accounts for a very small percentage of deforestation because these trees are most of the time replanted to sustainably harvest the area for a long time. Currently, the main threat to forests is agriculture as it induces significant deforestation either to grow crops or to cattle farms. Overpopulation, illegal tree cuts or wars are other important drivers of deforestation. (21)

In truth, in the past 30 years (from 1990 to 2020), the Earth has lost approximately 178 million hectares of forests. This trend is currently changing as net annual deforested area is still negative but less as it used to be. This is thanks to reforestation initiatives taken, for example, by China. At the moment, South America and Africa are the 2 continents suffering the most from deforestation. (22)

A tool which can help forests around the world and timber to be harvested sustainably is labels. In fact, labels like PEFC or FSC allow to guarantee the legal origin of the wood and

its sustainable production. The first one was created in 1997 and is more common in Europe and Belgium. The second one was created in 1993 and is focused around 10 main management principles. Here are some of the criteria which need to be respected in order to obtain the PEFC label: Strict limitations on forestry conversions (primary forests cannot be altered), no chemicals authorised, ecologically important forests must be protected, local consultations, etc. (17) (24)

Hereunder, in Figure 8, we can observe the countries members of the PEFC organization whereas in Table 3, we can observe the Countries or Regions accounting for the most part of PEFC labelled forests. As we can see, many countries are still not part of the organization and member countries don't have all of their forests certified meaning that progress can still be made. In total, 297 million hectares are currently PEFC certified. This represents only 7.3% of the forests of the world.

Climate change also has an impact on the forests of the world. Assuredly, the changes in temperature are affecting the forests. As winters are less cold, insects are not killed and therefore keep proliferating. As summers are warmer, droughts are more and more common making the trees more sensitive to insect infestations, forest fires and so on. (24)

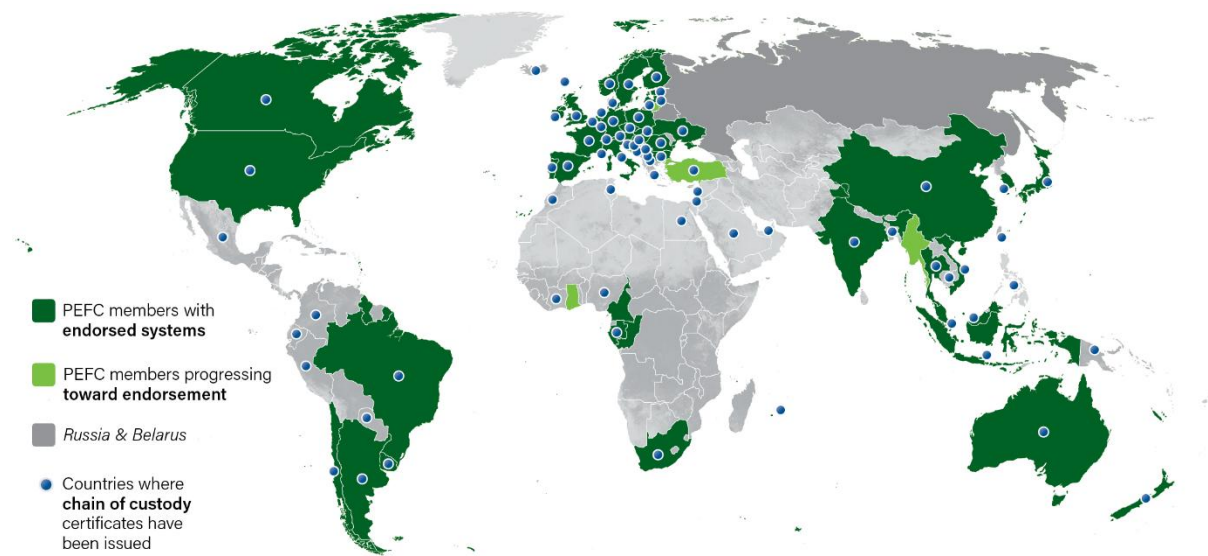


Figure 8: Countries members of the PEFC. Source: PEFC.

		Certified area [M ha]	National/Regional coverage [%]	Percent of total
1	Canada	133.3	38.4	44.9
2	United States	33.2	10.7	11.2
3	Finland	19.5	87.0	6.6
4	Australia	17.2	12.8	5.8
5	Sweden	16.4	58.6	5.5
6	Germany	8.9	78.3	3.0
7	Poland	7.3	76.6	2.4
8	Norway	7.3	59.6	2.4
9	Malaysia	6.7	35.1	2.3
10	France	5.7	33.2	1.9
	Total	255.4	490.4	86.0

Table 3: Top 10 Countries/Regions accounting for the most PEFC certified forests in the world. Source: PEFC.

Finally, in Figure 9 we can observe the increase in production of roundwood. Interestingly, the growth of firewood for the sole purpose of being burnt to recuperate the energy or to heat up is less strong as the wood grown for industrial purposes. This trend is beneficial as industrial wood has a better environmental impact than directly burnt wood. It is also expected that the growth of production of roundwood will increase significantly in the next decades. In Figure 10, we can observe some of the products achievable through industrial wood.

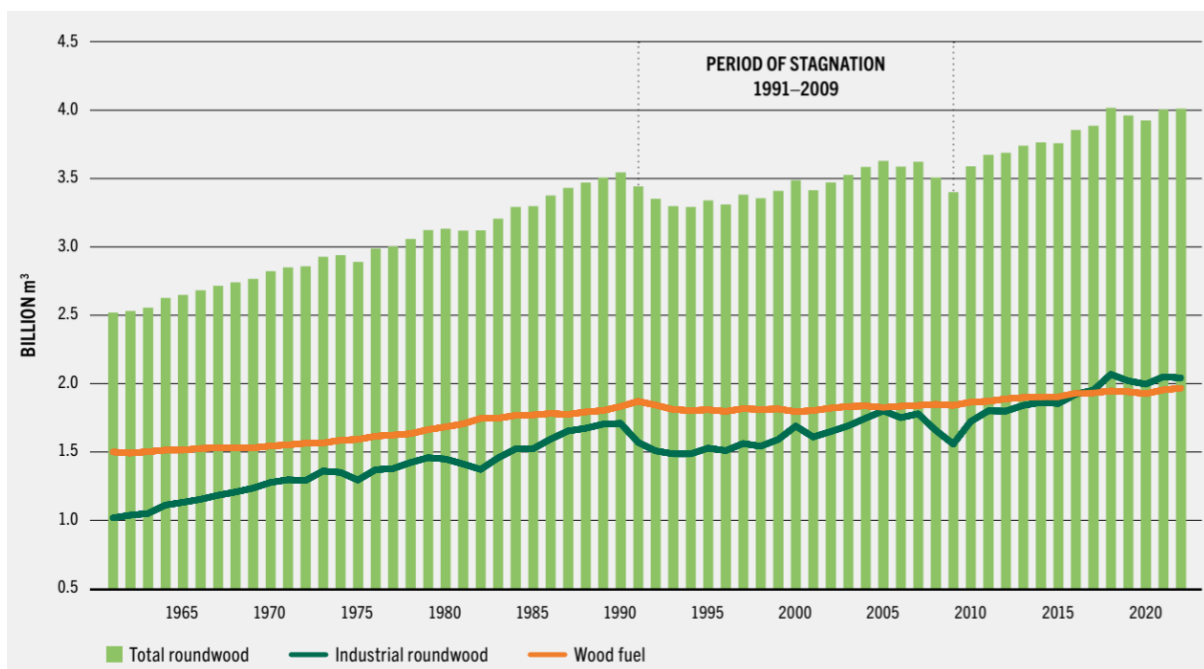


Figure 9: World roundwood production comprising industrial roundwood and wood to burn. Source: FAO.

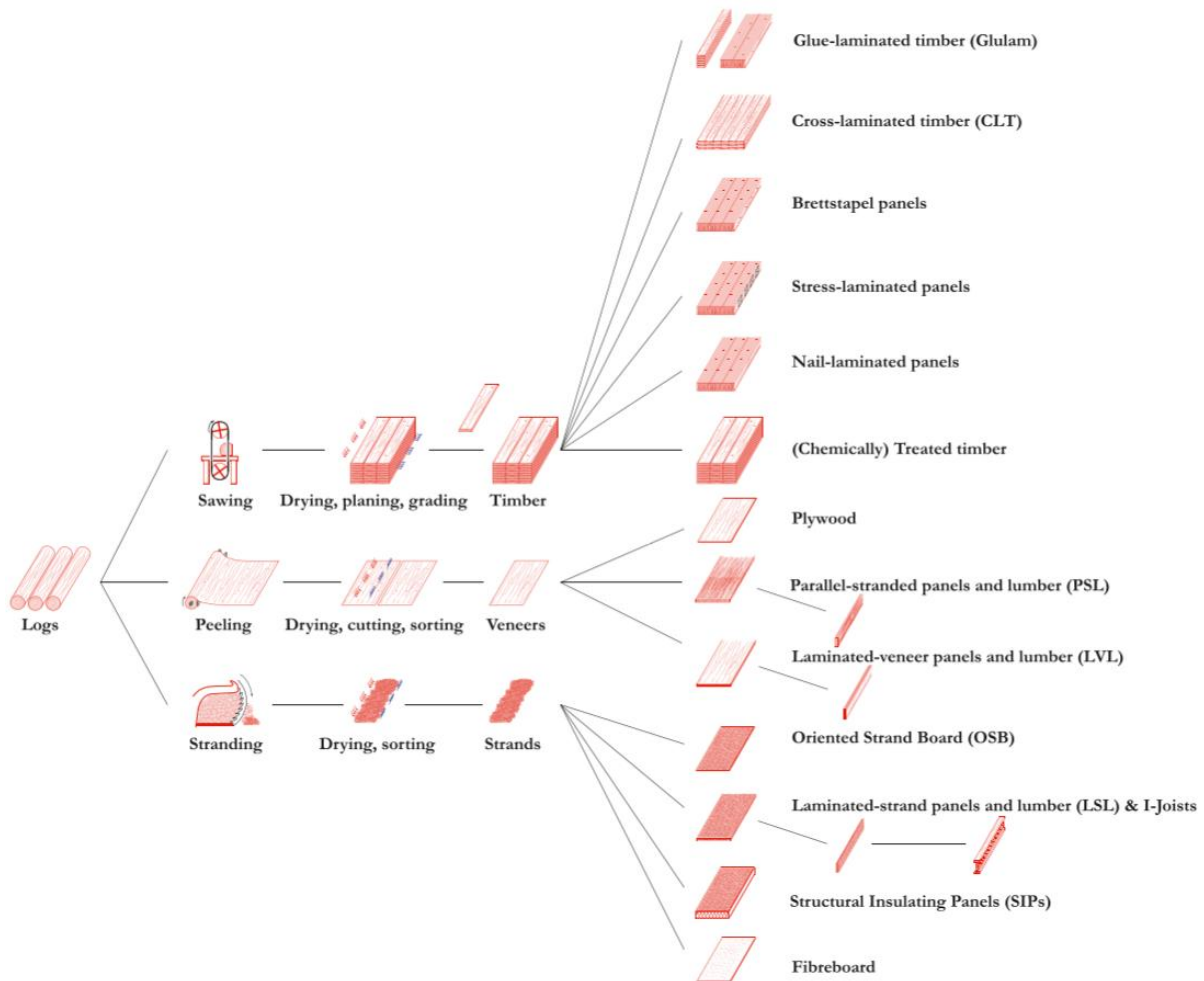


Figure 10: Achievable products from timber. Source: P.H. Fleming. (68)

In conclusion, the current threats to forests are not timber production for construction but agriculture and climate change. The massive use of sustainably produced timber to use for construction purposes can have a beneficial impact on the forests of the world as they would be better taken care of. Additionally, if harvested wood is initially used for construction purposes, then subsequently repurposed in another construction before being incinerated at the end of its life cycle, then the result is an exceptionally efficient use of the material. (20) (25) In the RoboTronc project, as wood is not significantly transformed, it can more easily be re-used in its original or in a transformed state for future projects. Finally, in order for forests to proliferate, forest managers must equalize harvested volumes to growth, maintain the biodiversity, productivity and regeneration capacity of the forests as well as making sure that the forests keep their ecosystem services. (26)

1.5. The complex properties of wood

Key aspects of the section: Wood stands apart from steel and concrete due to its organic nature, anisotropic structure, and variability in properties. It possesses a specific shape and complex internal composition. Wood's hygroscopic behaviour causes dimensional changes with moisture impacting assemblies and finishes. Strength and rigidity depend on species, grain direction, and moisture content. Defects like knots and cracks also affect its properties. Some of its key properties will here be modeled numerically in the following sections.

What differentiates wood from its main opponents: steel and concrete? A very small summary about the properties of wood and what makes it a complex but interesting material to work with is here depicted.

To begin with, as wood is an organic material, its shape will depend on many parameters. Its final shape when being cut will resemble the one of a truncated cone. This is an important characteristic of the material here considered for the RoboTronc project as it has many consequences on the assemblies, which we will discuss more extensively later on. In reality, as we can see in the Figure 11, the average cross-section of the tree reduces with the height of the tree. Additionally, the sections of a log are almost never perfectly circular. There can be excrescences. Moreover, if we draw a line between the centre of the bottom and top section, we can see that most of the tree trunks have their medium fibre off-centre.

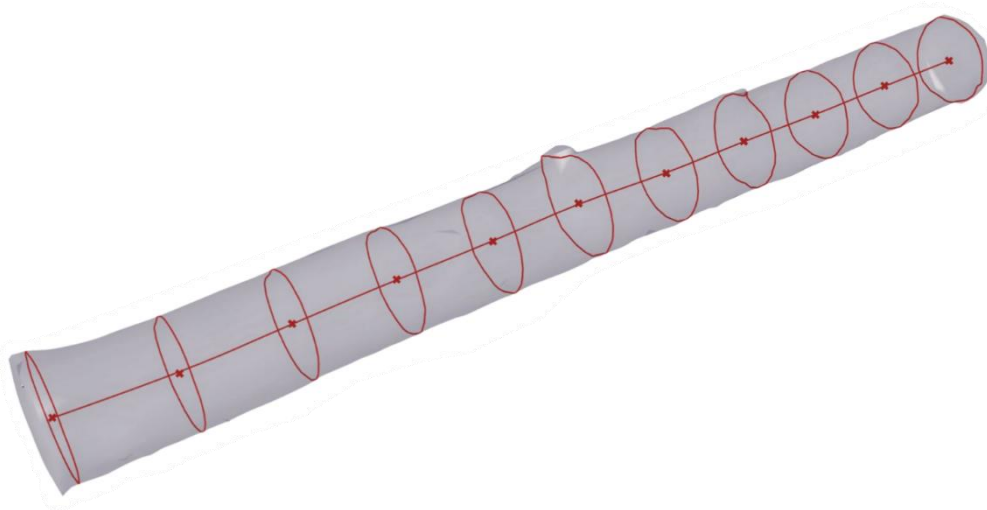


Figure 11: Shape of a log with its sections and its medium fibre. (3)

On top of a peculiar global shape, wood also has a sophisticated internal structure as we can see in the Figure 12. We can see that trees are composed of annual rings, fibres, cell walls, fibrils and molecules.

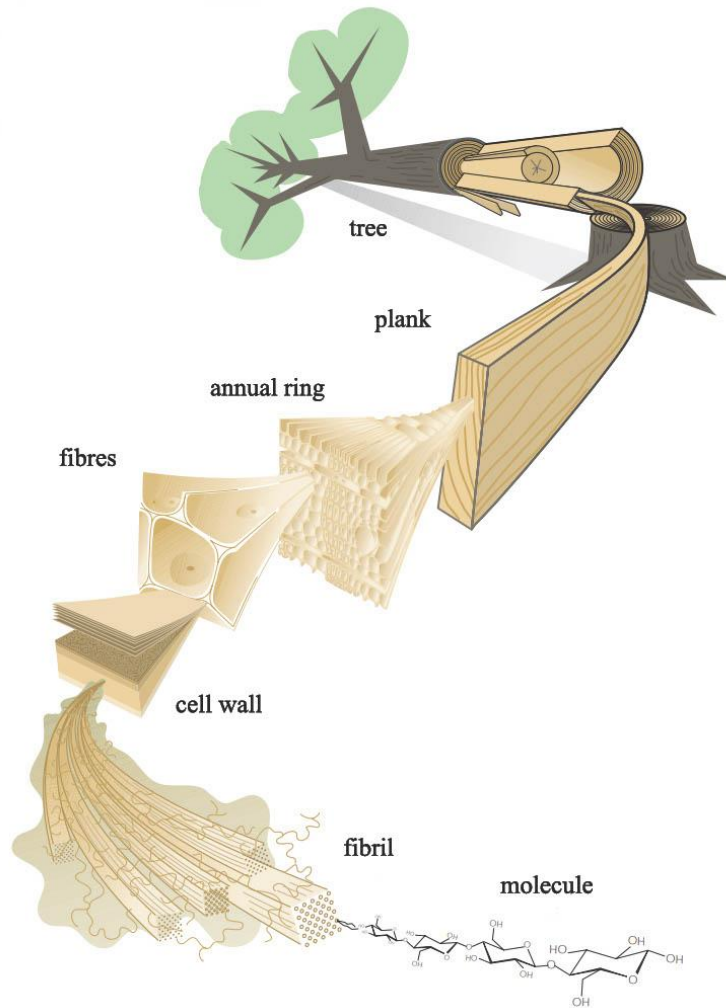


Figure 12: Internal structure of a tree. Source: adapted from Airi Illiste.

If we focus on a section of tree, there are 3 main areas visible to the naked eye (see Figure 13): the marrow, the heartwood and the sapwood. The first one corresponds to a cylinder of small diameter at the centre of the tree or a branch. The second one, the heartwood, has the role to mechanically support the tree. Therefore, it's the area of the tree with the best durability. The last one, the sapwood, is a few centimetres thick and is constituted of alive cells recently formed. It is commonly sensitive to insects and mushrooms but easily impregnable by treatments. It is globally advised to remove it to use the timber for structural purposes or long-term. Finally, there is also the cambium, phloem and, of course, the cork protecting the outside of the tree.

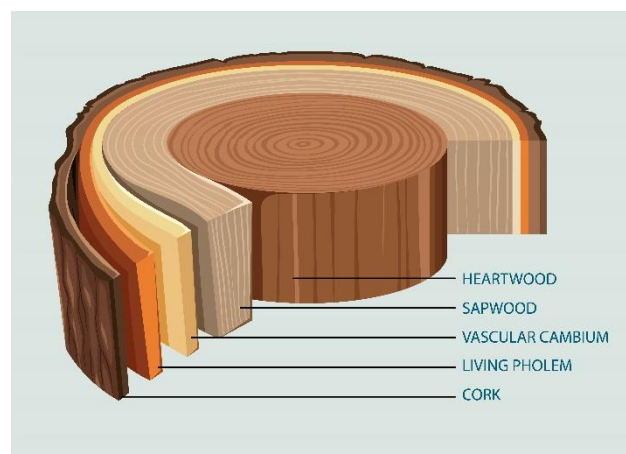


Figure 13: Layers of a tree trunk. Image by brgfx on Freepik.

On a chemical point of view, trees are mostly composed of carbon (50%) and oxygen (43%), the rest being hydrogen

(6%), nitrogen and diverse minerals. They combine to form three main types of macropolymers: cellulose, hemi-cellulose and lignin. Additionally, there exists two main types of trees: softwood (coniferous) and hardwood (deciduous). Softwood trees come from gymnosperm seeds where hardwood trees come from angiosperm seeds. As hardwood trees have a slower rate of growth, they are usually denser and therefore stronger. The way their nutrients flow and their precise internal structure also differ from the first type to the second. See (17), (21) and (27) for more information's on the topic.

Due to the fact that wood is an organic material, it also has defaults such as knots. Knots are considered to be a hole in the material when the timber is in tension and a hard point when in compression. They can be problematic as they can act as concentration points for the stresses. Another important and common default of timber elements are cracks. They are most of the time induced by changes in moisture content or by high stresses. The moisture content of wood is described first.

First of all, freshly cut trees contain a lot of water. If the wood is intended for construction purposes (or even burning purposes), it is usually advised to dry it either passively by storing it in a dry and ventilated environment or actively through the help of kilns for example. The first technique requires from 6 to 12 months or more whereas the second one requires around 6 to 8 weeks but obviously consumes electricity. Wood contains three main types of water. The first “type of water” contained in wood is called “**free water**” and it is contained inside the macro-void structure. This type of water interacts with wood through surface forces. When wood loses this water, no changes of volume are visible because the water is simply in the pores. Secondly, “**bound water**” or “**cell wall water**”, is a type of water in the wood which closely interacts with the cell wall polymers: lignin, hemicellulose and cellulose. It can be extracted from the wood by passive or active drying but it impacts the intrinsic properties of wood: volume, strength, conductivity, etc. Finally, “**combined water**” makes the last type of water that can be found in wood. This one is chemically linked to the structure and cannot be extracted without altering and destroying the wood. (17) (28)

Once dried to a certain extent, wood will still interact with the water present in the environment either in a liquid form (rain) or a vapour form (humidity). This is a crucial part of the knowledge required to build using wood as the changes in water content will induce significant changes in the dimensions of the material therefore affecting the assemblies, finishes, etc. The following key terms and definitions are explained. (17)

- **Water content (of air):** expressed in $[g/kg]$, it represents the mass of water vapor per kg of dry air.
- **Absolute humidity:** expressed in $[g/m^3]$, it represents the mass of water vapor per m^3 of humid air.

- **Absolute humidity of saturation:** also expressed in $[\text{g}/\text{m}^3]$, it is dependent on the atmospheric pressure and the ambient temperature. In fact, it represents the maximum mass of water vapor that $1[\text{m}^3]$ can hold. Following the Mollier diagrams, a transformation of vapor water to liquid water will happen if the air gets colder, if the water contained in the air increases or if pressure rises.
- **Relative humidity, ϕ :** expressed in $[\%]$, it represents the ratio of absolute humidity to absolute humidity of saturation for a certain atmospheric pressure and temperature.
- **Water content of the wood, w :** is the value most commonly used to assess the water level of the wood considered. It informs on the exact quantity of water contained in the wood following the Equation 1. The anhydrous mass corresponds to the mass of the wood without its free water and bound water. This value can be greater than $100[\%]$ for some species. In fact, a w of $200[\%]$ signifies that the mass of water equals two times the mass of dry wood.

$$w [\%] = \frac{\text{Humid mass of wood} - \text{Anhydrous mass } M_0}{\text{Anhydrous mass } M_0}$$

Equation 1: Calculation of the water content of wood.

The water content of wood can be measured through devices like a hygrometer which measures the resistivity of wood between two electrodes embedded in the wood. Depending on the future location of the timber, there exists recommendations of water content. For example, in Belgium, if the timber is aimed to be used in an outside environment, it is advised to dry it up to approximately $w = 16[\%]$. Whereas when aimed to be used inside, a w of $10[\%]$ is usually appropriate. This value aims at reducing the changes in dimensions which will appear when the wood will have increases and decreases of water content throughout the seasons. It is also highly necessary to reduce the risks of insect infestations or the propagation of fungi. (17)

Speaking of variations of dimensions, it is important to know that they vary depending on the direction of the material. Wood has three mutually perpendicular directions in which it will react differently structurally as well as vary in dimensions with changes of water content. Therefore, we can say that wood is orthotropic as we can see in Figure 14. The three main axes being the **longitudinal**, the **radial** and the **tangential** axes. Three interesting

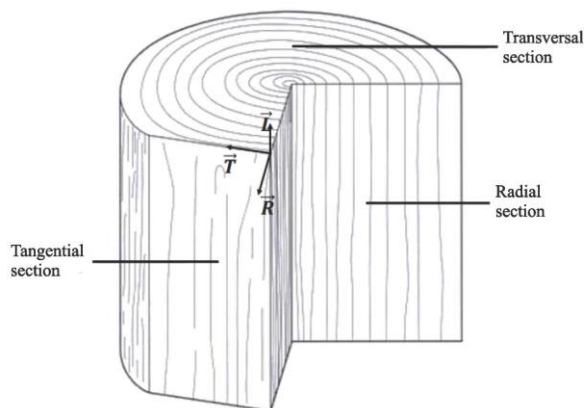


Figure 14: Orthotropic axes of wood together with the 3 main sections.

sections also arise: the **transversal**, the **radial** and the **tangential** sections. Thanks to the Equation 2 and Table 4, we can now calculate the variations of dimensions of wood in the three main axes of the material.

$$\Delta L = \frac{\beta}{100} \cdot \Delta w \cdot L$$

Equation 2: Variation of dimension of wood with water content.

With:

- ΔL : the variation of dimensions in [mm] in the selected direction.
- β : the average shrinkage or swelling coefficient expressed in [%/%].
- Δw : the variation of water content of the wood.
- L : the initial dimension in [mm] in the selected direction.

	β: Average shrinkage/swelling coefficient expressed in [%/%]		
	Tangential	Radial	Longitudinal
Spruce	0.27	0.13	0.01
Scotch pine	0.28	0.17	0.01
Larch	0.27	0.14	0.01
Beech	0.39	0.19	0.01
Oak	0.32	0.15 – 0.19	0.01
Douglas	0.23	0.16	0.01

Table 4: Average shrinkage/swelling coefficient β of key species. (17)

Additionally to changes in dimensions due to changes in water content, wood is also impacted by thermal changes as we can see with the Equation 3 and Table 5.

$$\Delta L = \alpha \cdot \Delta T \cdot L$$

Equation 3

With:

- ΔL : the variation of dimensions in [mm] in the selected direction.
- α : thermal dilatation coefficient expressed in [mm/mm/°C].
- ΔT : the variation of temperature of the wood.
- L : the initial dimension in [mm] in the selected direction.

		α : thermal dilatation coefficient expressed in [$10^6 \cdot \text{mm}/\text{mm}/^\circ\text{C}$]
Wood	Tangential axis	25 – 60
	Radial axis	15 – 45
	Longitudinal axis	3 – 6
Steel		12
Concrete		12

Table 5: Thermal dilatation coefficient α of wood in its 3 main directions compared to steel and concrete. (17)

Lastly, for this chapter, the strength properties of wood are reviewed. As discussed previously, wood behaves differently depending on the axis considered. First of all, it is important to note that wood, as a natural material, has a broader range of variability between species as well as between specimens of the same species compared to concrete or steel. The following tables representing strength parameters linked to wood were performed on relatively small specimens, clear of imperfections and on an extensive sample base. The RoboTronc project is also considering this challenge regarding the strength determination of wood by considering putting in place a non-destructive testing procedure of all the logs to be used in a project to make sure of their strengths. Interested readers are also invited to explore the past research (4) (6) (7) (29) as well as the future studies to come regarding that topic.

In order to describe the elastic properties of wood (30), twelve constants are required: three moduli of elasticity E , three moduli of rigidity G and six Poisson's ratios μ . The Equation 4 describes the general relation found between the moduli of elasticity and Poisson's ratios (however not always closely met for all species).

$$\frac{\mu_{ij}}{E_i} = \frac{\mu_{ji}}{E_j}, i \neq j, i, j = L, R, T$$

Equation 4: General relation between moduli of elasticity and Poisson's ratios.

The three moduli of elasticity are E_L , E_R , E_T and correspond, respectively, to the elastic moduli in the longitudinal, radial and tangential direction. Most of the time obtained through compression tests, they vary within and between species as well as with moisture content. Then, Modulus of Rigidity or “shear modulus”, indicate the resistance to deflection of a member caused by shear stresses. They are: G_{LR} , G_{LT} , G_{RT} and correspond, respectively, to elastic constants in the LR, LT and RT planes. For example, G_{RT} is the modulus of rigidity based on shear strain in the RT plane and shear stresses in the LR and LT planes. Finally, the Poisson's Ratio informs us on the deformation perpendicular to the direction of the load when a member is loaded axially. This deformation is proportional to the deformation parallel to the direction of the load. They are denoted as follow: μ_{LR} , μ_{RL} , μ_{LT} , μ_{TL} , μ_{RT} , μ_{TR} . The first letter of the

subscript denoting the direction of applied stress and the second one, the direction of lateral deformation. Just like the others, these parameters depend within and between species and are affected by changes in water content. The following tables (Table 6 and Table 7) highlight Elastic ratios of key species as well as the Poisson's ratios of key species. (30)

Species	E_T/E_L^{10}	E_R/E_L	G_{LR}/E_L	G_{LT}/E_L	G_{RT}/E_L
Spruce (Sitka)	0.043	0.078	0.064	0.061	0.003
Red pine	0.044	0.088	0.096	0.081	0.011
Larch (Western)	0.065	0.079	0.063	0.069	0.007
Birch (yellow)	0.050	0.078	0.074	0.068	0.017
Oak (red)	0.082	0.154	0.089	0.081	-
Douglas	0.050	0.068	0.064	0.078	0.007

Table 6: Elastic ratios of key species at approximately 12% water content. Adapted from (30).

Species	μ_{LR}	μ_{LT}	μ_{RT}	μ_{TR}	μ_{RL}	μ_{TL}
Spruce (Sitka)	0.372	0.467	0.435	0.245	0.040	0.025
Red pine	0.347	0.315	0.408	0.308	-	-
Larch (Western)	0.355	0.276	0.389	0.352	-	-
Birch (yellow)	0.426	0.451	0.697	0.426	0.043	0.024
Oak (red)	0.350	0.448	0.560	0.292	0.064	0.033
Douglas	0.292	0.449	0.390	0.374	0.036	0.029

Table 7: Poisson's ratios of key species at approximately 12% water content. Adapted from (30).

The parameters cited above will be useful to the numerical model later considered in this paper. But most of the time, the engineers will not use these parameters but instead, more directly applicable values such as the modulus of rupture, compressive strength parallel to grain, perpendicular to grain, shear strength parallel to grain as well as tensile strength parallel and perpendicular to grain. It is important to note that strength and rigidity is highly impacted by the water content of wood. A high water content induces much lower strength and rigidity.

¹⁰ With E_L , the modulus of elasticity in the longitudinal direction, described in Table 8.

Strength can approximately be divided in half when w is higher than 20%. (17) The average longitudinal modulus of Elasticity as well as the compressive and tensile strength (parallel and perpendicular to the grain) of key species is described in the Table 8. It is important to note that the moduli and strengths are high because these are average values and not characteristic values like used in the Eurocode (NBN EN 338/2016). (2)

Species	Longitudinal modulus of Elasticity [MPa](E_L)	Compressive strength parallel to grain [MPa]	Compressive strength perpendicular to grain [MPa]	Tensile strength perpendicular to grain [MPa]	Tensile strength parallel to grain [MPa]
Spruce (Sitka)	10 800	38.7	4	2.6	59.3
Red pine	11 200	41.9	4.1	3.2	80.0
Larch (Western)	12 900	52.5	6.4	3	111.7
Birch (yellow)	13 900	56.3	6.7	6.3	109.6
Oak (red)	13 100	48.5	7.8	-	112.4
Douglas	12 300	47.6	5.3	2.7	107.6

Table 8: Essential strength properties of key species at approx. 12% of water content. Adapted from (30).

In conclusion to this chapter, wood requires many parameters to try to get a good idea of its behaviour against stresses or changes of humidity in comparison to its main competitors like concrete or steel. This does not make it an unsuitable material for construction but requires a strong knowledge for the designers and builders. There is of course a lot more to discuss about on this topic and we invite therefore the readers interested in that area to reach for the corresponding sources cited in this paper. (8) (17) (21) (30) (31) (32)

Chapter 2

Numerical simulations of wood



Icon from Uniconlabs.

2. Numerical simulations of wood

When designing a structure, one can start analytically by writing on a sheet of paper. Nowadays, structures have become more complex and need more optimisation making it impossible for reasonable analytical solutions. Therefore, engineers have moved towards a numerical solution allowing them to simulate, predict and understand structures and materials.

This chapter two discusses why numerical simulations are essential today and for the future and in particular, to better understand the material wood. Firstly, a short state of the art summarises recent developments and current challenges in the interesting field of numerical simulations of wood. It is not exhaustive and therefore the reader is invited to reach for the sources mentioned here. This will be followed by an introduction to one of the possible methods of numerical simulation: the Finite Element Method (FEM). The software chosen for this master's thesis, Abaqus 2021, will also be introduced and explained. Afterwards, the main material models to consider for wood are described and compared with the use of introductory exercises: a simply supported beam and an overhanging beam. Finally, key parameters of numerical simulations are discussed: the contacts between the elements, the tolerances and their consequences as well as the modelling of the non-linearities of wood further detailing the material model chosen. The final goal of this chapter is therefore to have a relatively simple material model numerically simulating wood. This numerical model will later be used in the Chapter 3 to replicate an experiment and analyse its strengths and weaknesses.

First of all, what are exactly numerical methods such as FEM? They are techniques trying to approximate complex solutions using a set of numbers associated with mathematical equations. Most of the time, they transform mathematical expressions into a set of algebraic equations depending on thousands or millions of unknowns, therefore requiring to be solved by computers. Interestingly, numerical methods are strongly linked and inseparable to three main categories: mathematics, computer science and material modelling. Therefore, what should a structural model include to work? (33)

- **A geometric description** of the structure using points, surfaces or volumes.
- **The properties of the materials** in the structure and their behaviour with regards to external changes: structural solicitation, changes of hygrometry, etc.
- **Mathematical foundations** describing the physical laws and governing the behaviour of the structure: force-equilibrium equations, boundary conditions, etc.
- **The forces** acting on the structure.

On this structural model, simulations can be performed once a numerical method is chosen. There exist four main methods: Finite Volumes, Finite Differences, Particles Methods and Finite Elements Methods or FEM. The last one is the one chosen here. The numerical method must also be fed with the above-mentioned aspects as well as a discretization: element type, mesh size, etc. The final outcome being a **computational model** for the analysis of a structure. A summary of the different aspects is visible in Figure 15.

As previously mentioned, numerical simulations have become indispensable as the complexity of calculations and structures surpasses the capabilities of analytical solutions. While experimental campaigns remain valuable for verifying and deepening our understanding of phenomena, they are time-consuming and resource intensive. Numerical simulations, although requiring significant research—particularly for wood—offer the potential for fast, cost-effective, and, ideally, accurate analyses. Moreover, they allow data extraction from any point within the material, which is not feasible with experimental testing. The following subsection highlights recent advancements in the numerical analysis of wood.

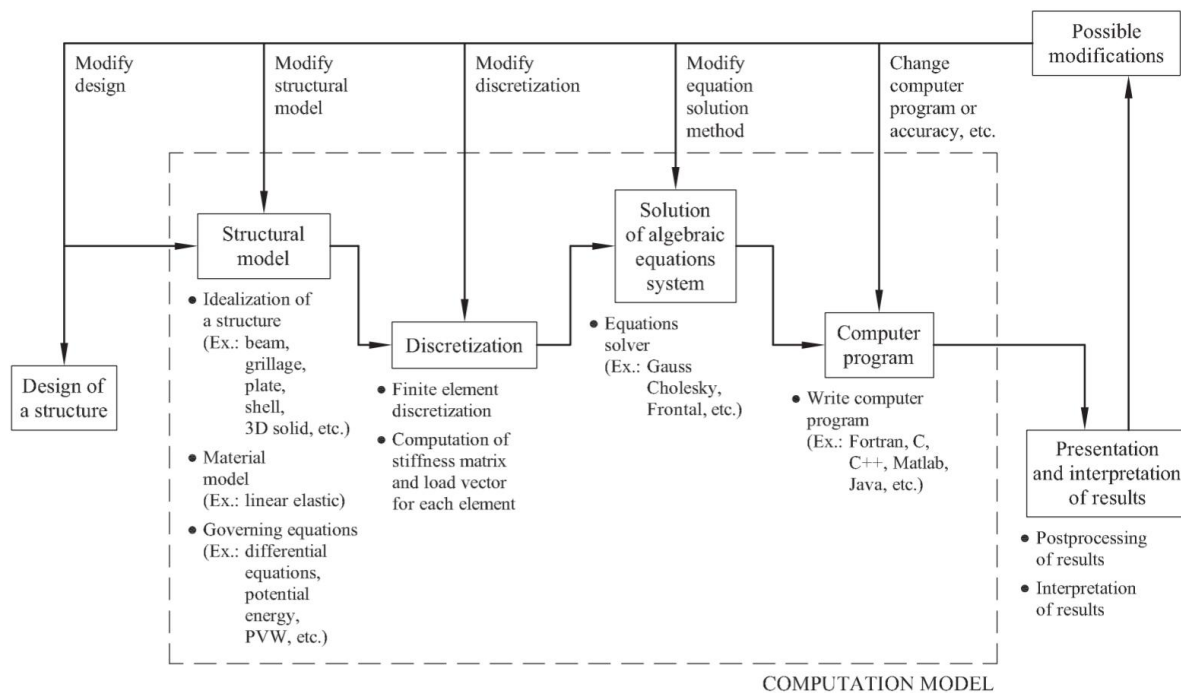


Figure 15: Scheme of the FEM for structural analysis. (33)

2.1. A short State of the art of Numerical Analysis of wood

Key aspects of the section: Ongoing research in the numerical analysis of wood aims to model its behaviour under various loads across countless scenarios, providing a deeper understanding of the material, optimizing structures, reducing costs, and enhancing safety. To achieve this, researchers are exploring the wood's microstructure, the effects of knots, mesh selection, element deletions, Cohesive Zone Models (CZM), and fibre crushing, among other factors. Their results are often compared with experimental data. While the findings are promising, significant numerical challenges persist and the complexity of the models is most of the time inaccessible to an engineer without a strong background in the field of numerical analysis of wood. This master's thesis will try, in the following sections, to make it accessible and reproducible.

Introduction

As numerical analysis of wood is a complex task being constantly revised, it is of utmost importance to first read extensively past techniques and advancements in order to visualize the different possibilities, current challenges and needs. In fact, there exist many methods and levels of complexity and they need to be carefully chosen for a specific project. Some of the latest advances in numerical analysis of wood are here briefly described to have a broad view of the possibilities and challenges.

The reasons why it is so hard to make a global and ready-to-use model working for wood are multiple: strong anisotropy of the material, heterogeneity, ductility in compression but brittle behaviour in tension and shear. The material showcases brittle failures in tension parallel to the fibres and plastification in compression perpendicular to the fibres. There is also a strong variability between species and between specimens inside a same species. It also varies with age, knots, cracks, natural deformations or moisture content. Therefore, the models used have limitations and must be adapted to the situation with the use of parameters.

The parameters considered for a numerical simulation with wood can greatly vary depending on the chosen complexity of the model as it will be described in this section. From structural parameters (Young moduli, rigidity coefficients, Poisson ratios) to moisture parameters, temperature, etc. To acquire parameters from the exact logs to be used for construction, research has been ongoing regarding non-destructive tests. (4) (6) (7) At the moment, these non-destructive tests can extract the key structural parameters mentioned above (namely the Young's moduli, rigidity coefficients and Poisson ratios) with relative precision. The longitudinal Young's modulus being the most precise parameter obtained. Research is still required to increase efficiency and quality of the results.

The possibilities of numerical analysis are plentiful. In fact, there exists tools to scan the exact geometries of logs and to have parametric computerized millings implemented in the software to make assemblies. Photogrammetry is one of the tool that can be used for the scans (although not practicable in industry), Metashape (MS) to build the numeric model and Rhino with Grasshopper to build parametric cuts into the elements to be later performed by robotic armed and assembled together. (3) There also exists easier-to-use scanning devices like the ones already used by most professional sawmills: Faro Focus, Hexagon Leica BLK360, Trimble X7 and many more.

Another possibility of numerical analysis is to perform numerical structural tests with the help of software's like Ansys or Abaqus. This can allow to visualize the stresses in the material through time and depending on the loads applied. Deformations are also extractable as well as the variation of many other parameters: hygrometry, volume, etc. A significant challenge to achieve that is to accurately model the material wood and its behaviour with time and against loads in different directions. As visible in the literature, researchers often combine numerical simulations with experimental campaigns and analytical results (when possible). This allows for comparisons as well as calibration of the models.

Fracture modelling

For the past decades, many researchers like Cabrero et al., Akter et al. or Serrano have experimented with failure criteria, like the ones described in Annex 5.1, to compare their results. Serrano has recently (2020) experimented with a simplified plasticity model for wood in compression. One of the challenges they encountered is that wood-wood connections have a discontinuity in the fibres. Therefore, forces transferring where cracks are located can propagate such cracks and induce fractures in the wood parallel to the fibres. For wood, three fractural modes are identified as visible in the Figure 16. They correspond to a tendency of a crack tip stress to decompose into a fracture. (34)

- Mode I: due to stresses orthogonal to the local crack plane surface.
- Mode II: due to stresses parallel to the local crack plane surface but orthogonal to the crack tip surface.
- Mode III: due to stresses parallel to the local crack plane surface and crack tip surface.

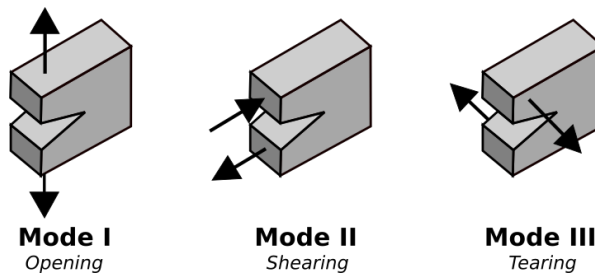


Figure 16: The three fractural modes. Source: fracturemechanics.org.

Consequently, a strength factor can be used to express the strength of a timber specimen to resist to fractures: the Stress Intensity Factor or Fracture Toughness, K . It represents the highest stress intensity that the material can withstand before crack propagation. Regarding crack propagation, Cohesive Zone Models (CZM) can also be implemented with the help of FEM. They simulate the initiation and propagation of cracks and other types of failures. The fracture is modelled by describing the mechanical behaviour of a small region ahead of the crack tip called the cohesive zone. To model the process of damage accumulation and crack propagation, the CZM uses a traction-separation law governing how stresses and displacements evolve in the zone of interest. Koch et al. (2013) have experimented tapered tenon joints with CZM with the help of FEM to increase the knowledge of failure modes I, II and III of lamella separation. They found discrepancies with experimental failure capacities and believe it is linked to the influence of friction coefficients. (34)

Numerical modelling of scarf joints

Sangree and Schafer (2009) experimented on scarf joints using 3D linear elastic FE models and used a variation of material models: isotropic, transversely isotropic and orthotropic as well as models to predict the contact and stiffness with respect to gaps. They also studied the impact of stresses relative to the direction of the fibres. Other researchers like Guan et al. (2008) decided to experiment with 3D nonlinear FE models considering an orthotropic elastic material model in tension and an elasto-perfectly plastic model in compression. (34)

As we can see, a significant challenge in numerical analysis of wood is to properly simulate the behaviour of the material. Most of the time, wood-wood joints will be simplified, and a common approach is to use an orthotropic material model with transverse isotropy. This method being simpler, it is more accessible to an engineering study and allows for gains in computational cost as well. (34)

User subroutines in Abaqus

Alternatively, some authors have gone the other direction by trying to push the complexity of their models to be as close as possible to reality. In order to do so, software's

like Abaqus for example, allow to write “User Material Code” or “UMAT” code to incorporate in the software material properties and failure criteria. Many other parameters can also have a significant impact on the results of the simulations: element selection, mesh concentration, degrees of freedom, friction coefficients, contact surfaces, etc. A common type of element used in Abaqus is C3D8R. Another common method to improve results is mesh densification in areas of high interest (also called “h method”). (34)

Experimental tools to compare with numerical results

So far, researchers have managed to relatively well simulate the material wood but at the cost of complex methods and techniques requiring detailed knowledge in the field. This has also allowed to increase the understanding of the material and the behaviour of specific connections especially for zones in the material which are inaccessible to real-life experimental measuring tools. A tool commonly used for experimental campaigns is called “Digital Image Correlation” or DIC, and it allows to track stress distributions and crack propagation but, so far, there is limited research where this tool has been used for FEM validation.

Similarly, high quality strain gauges like optic fibre sensors haven’t yet been investigated much to validate FEM but research is ongoing in this field. Most of the time, FEM are validated based on the outcome of failure modes. (34) A tool used during experimental campaigns to try to validate FEM is “ARAMIS-System”. This tool is a commercial system implementing DIC and providing specialised hardware such as cameras and lighting as well as a specific software to measure, analyse and visualise deformations using DIC. As visible in Figure 17, the results obtained experimentally using ARAMIS-System can approximately be compared to the Numerical results obtained using Abaqus. (35)

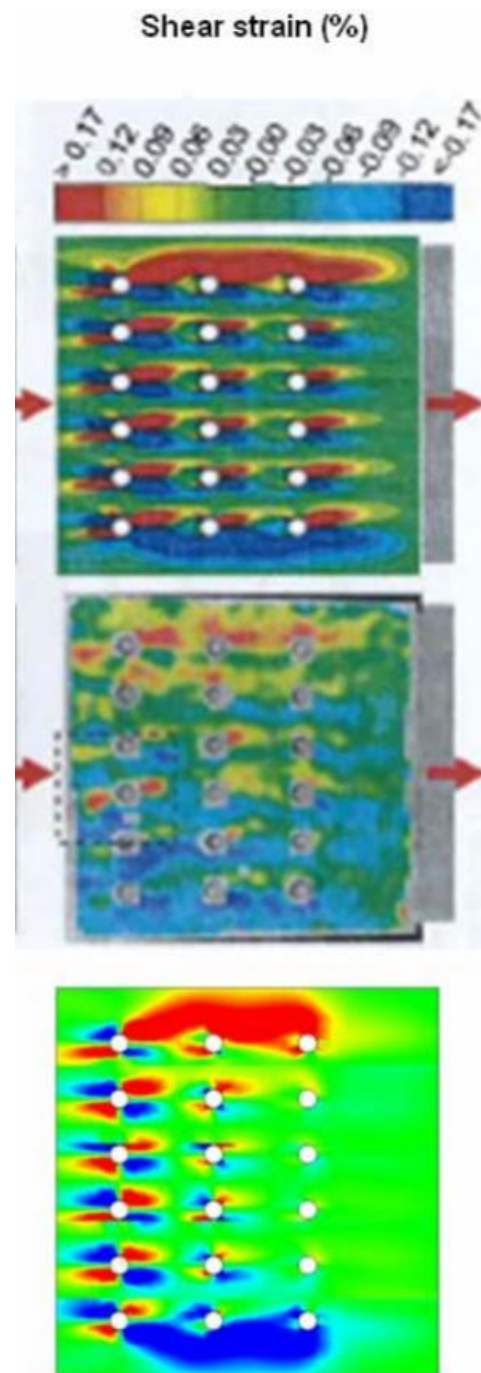


Figure 17: Comparison of experimental results using ARAMIS system with Numerical results using Abaqus. (35)

Alternative use of numerical analysis of wood

Additionally, Finite Element Analysis (FEA) can also be pertinent in assessing the resistance of assemblies of existing buildings to assess if reinforcements are needed or not. Voulpoitis et al. (2021) have proposed a framework to evaluate the structural strength of tall timber buildings. (34)

Parametric modelling

Another interesting field of research among numerical analysis of wood is regarding the emerging use of parametric software's like Rhino/Grasshopper. In fact, the parametric possibilities allow for quick changes to the structure: number of elements, orientation, etc. It also allows for changes regarding the geometries of the assemblies themselves. Additionally, numerical simulations can also be performed using a plug-in called "Karamba3D". It has the advantage of performing all the design and dimensioning in the same interface but so far, the numerical simulations are limited in possibilities compared to more professional software's like Ansys or Abaqus. In fact, Karamba3D is limited to beam, truss and shell structures and is not designed for 3D solid analyses. Moreover, beam elements are limited to isotropic material properties and orthotropic is only currently limited to shell elements. Therefore, the software's allows for quick simulations but limited in accuracy. (8)

Dupriez and Stevens (2022) have in their research analysed the natural frequencies and deflexions of whole logs to be used as beams. Their numerical results were compared to experimental results obtained by Houben (2022). The results obtained showed comparable results. As they performed all their tests on Karamba3D as well as Abaqus, results allow us to compare an isotropic model (Karamba3D) to an orthotropic and more sophisticated model (Abaqus). Interestingly, the isotropic results generally show less deflection than the supposedly more accurate orthotropic model and therefore, users of isotropic models must be careful that the expected deflection is higher. This however requires more research and a comparison with an extensive experimental campaign to assess the real-life deflection.

Numerical analysis of timber logs

The literature of numerical analysis of wood mostly focuses on squared timber or engineered timber as they are the most common types of wood used in construction. However, research is emerging regarding whole logs used as columns or beams. In this matter, sometimes the logs will be approximated as simple cylinders, truncated cones or their whole complex geometry will be taken into account in the software's. Algorithms have been developed to identify the medium fibre of real tree trunks as their geometry is most of the time complex. (3) (8) What has also been studied is the influence of the angle of notches on logs to assemble them together. They found that a notch of a depth of 30% of the radius of the log will endure an

increase of 10 to 15% in the global deflection of the beam. And obviously, deeper notches degrade even more strongly the bending strength of the beam. Moreover, when the notch depth is close to the centre of the beam, a possible concentration of the stresses can happen at the location of the notch. Just like for any numerical analysis on wood, the values of the parameters describing the material are essential and have great consequences on the results. Accurate parameters adapted to the wood species chosen is primordial. (8)

Numerical analysis of the impact of moisture in wood and other parameters

Research has also been ongoing regarding the impact of the changes in moisture content in wood through time. Whereas we will not dig into this field in this master's thesis, it shows the wide range of work in the field. Mirianon et al. (2008) (35) have been using Abaqus and DFLUX user subroutines to study such changes through time and through different loading as well as joint geometries. Like usually, a combination of theory, numerical simulations together with experiments were conducted. In this research, they have used a staggering number of 39 parameters to describe the wood as closely as possible: Elastic moduli, Poisson's ratios, Shear moduli, Coefficient of moisture expansion, Moisture content of wood, Density, Temperature, etc.

Numerical analysis of wood with steel

Mirianon et al. (2008) (35) have also conducted numerical simulations on timber joints mixed with steel dowels. This is highly interesting and complex as steel is a much stiffer material and therefore it is complex to join the two of them and their interaction in the software. This time, they have taken into account loading time, load cases and moisture content of wood. They have discovered that the variation of moisture content of wood can have a strong increase of the stresses in the wood, especially in the direction perpendicular to the fibres. Additionally, they have also performed the same tests but with glued-in steel rods as reinforcement. The conclusion of this numerical and physical experiment is that the tension is converted to compression around the reinforcements, increasing the strength of the material locally.

The changes of moisture content are to be controlled as they can induce cracks in the location of the glued-in steel rods as steel doesn't change in volume with the changes of ambient humidity. They have also found that not only the water content in wood is important to understand and monitor but also the gradient of moisture content as it can induce stresses perpendicular to grain, potentially splitting the wood. The EC5 describes the failure modes possible when wood is linked with steel rods. (2)

Numerically, they have made varying the number and location of the dowels and compared it to experiments. Their numerical results are in accordance with the experiments. The fact that they managed to partially reproduce numerically what is happening in real life is

very promising as Numerical Simulations can be relatively cheap and quick allowing to perform many of them to find an optimal design and optimising structural safety. The authors insist that their results are still on a qualitative basis and that more research is required to make this method more reliable and accurate as well as accessible to the general public.

Innovative methods to model wood

Another approach visible in the literature to try to model wood as closely as possible to reality is by trying to model its microstructure. In fact, instead of using the usual macroscopic properties and geometries of wood, the cellular microstructure of wood is being modelled and numerical tests are performed. (See Figure 18) The microstructure is here approximated by a regular honeycomb-type pattern and irregularities are here neglected. The author, however, suggests that future research should try to implement these irregularities as they can have great consequences on the behaviour of the model and allow to get closer to reality. (36)

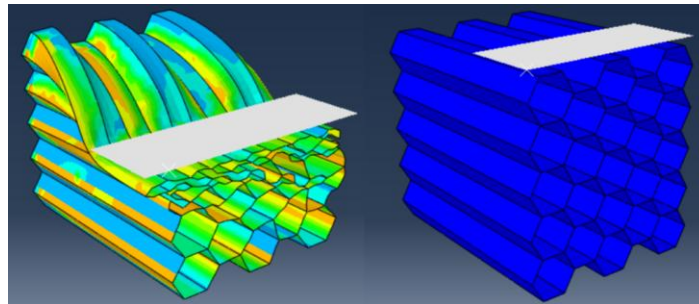


Figure 18: Numerical simulations on wood's microstructure. (36)

The cell structure is the reason for wood's anisotropic behaviour. Moreover, the macroscopic behaviour of wood is a combination of the cell wall behaviour and the cell structure behaviour. The microstructure is subjected to a transverse compression to numerically capture known physical effects like the Hammock effect¹¹ or the effect of increased stiffness for untouched areas. For this research, the author has decided to use Hashin damage model over other ones like Hankinson, Norris or Tsai-wu (as discussed in Annex 5.1). The honeycomb elements are here modelled to behave elastically and buckling of the cell walls is taken into account. An increase of the stiffness of the honeycomb structure is observed when cell walls collapse and touch each other.

Several challenges appeared like numerical instabilities appearing when deformation was becoming so large that many cells would cave in. Convergence difficulties also happened due to buckling and therefore, explicit dynamic techniques were used as they are more suited for complex contact conditions and buckling. In that case, it was difficult to assess if the solution found was close to the true static solution. The author also found that the time of

¹¹ The Hammock effect is a phenomenon where an additional strength can appear from the fibres acting like ropes or cables in tension (like a hammock) and holding up a deformed area from the sides. (36)

application of the load needed to be close to reality in order for the inertia forces not to dominate and not risk finding the quasi-static solution. However, this induces a high CPU¹² cost. Consequentially, the author found several ways to reduce CPU cost. To reduce the number of increments to reduce the total time span of the simulation. Increase the elements' size. Increase of the loading rates therefore reducing the number of increments.

A transversely isotropic lamina model was used as well as a displacement-controlled loading to reduce the risk of convergence difficulties and because the real-life load is unknown. The structure visible in Figure 18 is approximately 120[μm]-tall. The reaction force was the main information extracted from the tests and a resemblance to experimental data could be seen as the beginning of the test was showcasing a linear part as a start, a strain hardening region afterwards followed by a stiffness drop highlighting material failure and buckling of the top cell layer and ending with some hardening. Interestingly, when refining the mesh, unrealistic results appeared and this was discovered as the energy balance was very different from coarse mesh tests and the external work was less than the internal energy. Hence, a compromise needed to be found to capture the cell geometry but keeping energy balance and stability.

Several geometries have been tested but tests were limited to simple compression perpendicular to the fibres and they open the way to more types of tests. A significant challenge raised by the author is the passage from microstructure level to macrostructure level. However, if successful in future research, it could lead to highly precise simulations results managing to describe the material nonlinearities of wood and the failure modes. This could revolutionize numerical modelling of wood and especially the joints.

Numerical modelling of the compression perpendicular to fibres for wood elements

Research has also been ongoing to simulate wood under compression perpendicular to the fibres combined with a rolling shear. (37) In this research paper, the authors have experimented with a dog-bone shape in Abaqus software and varied the position of the annual rings of wood and compared the results to experiments. They considered three fracture criteria: Hill's criterion, Hoffman's criterion and QMS. The first one was found to be the least suitable to model the local material behaviour as well as the global structural response. This is because the Hill's criterion does not represent well the difference in compression and tension in the three principal direction. Whereas the other two models were in good agreement with experiments for biaxial tests. They have also experimented with the effect of the unloaded length and they observed a significant variation in design strength.

¹² CPU stands for Central Processing Unit. It corresponds to the key component of a computer to perform the tasks.

The authors modelled wood using Continuum Damage Mechanics (CDM). It corresponds to a theoretical framework to describe the gradual deterioration of the material under a load. It models how defects like cracks are forming and impacting the behaviour of the material. Practically, it can be taken into account by using a modified stiffness matrix where the stiffness parameters are reduced after failure initiation. Complex and hybrid models can also be imagined like using an elastic-plastic orthotropic beam element in combination with a crushable foam model in order to capture the plasticity linked to crushing of wood cells as well as the brittle failure related to the tension perpendicular to the grain and shear.

Promising results have been observed for multi-scale plasticity models but as they are not easy to follow and not readily available to engineers, the authors focused on a simple material model which they calibrated based on experimental data and applied to numerical simulations. They managed to represent the overall response of wood in the transverse plane for all loading cases apart from tensile loading. The influence of the annual rings' position was found to be more significant in the case of radial compression compared to tangential compression. The authors advise for more research in the field of hardening parameters for wood as it could improve its numerical modelling.

Numerical modelling of a wooden dog-bone with steel dowels

The next research paper (38) also used Continuum Damage Mechanics (CDM) to model the tension of a dog-bone shape together with dowels as visible in the Figure 19 as well as embedment tests as visible in the Figure 20. The authors tested three species of wood: spruce, beech and azobé and managed to identify the failure modes during their tests. The general shape of the load-displacement curves were mostly in agreement with the experimental outcomes. To identify the individual failure modes, they used the Hashin damage model described in the Annex 5.1. Therefore, their approach corresponds to a general constitutive model based on CDM in a single 3D constitutive model to describe brittle failure models in tension and shear together with nonlinear elastic failure modes in compression.

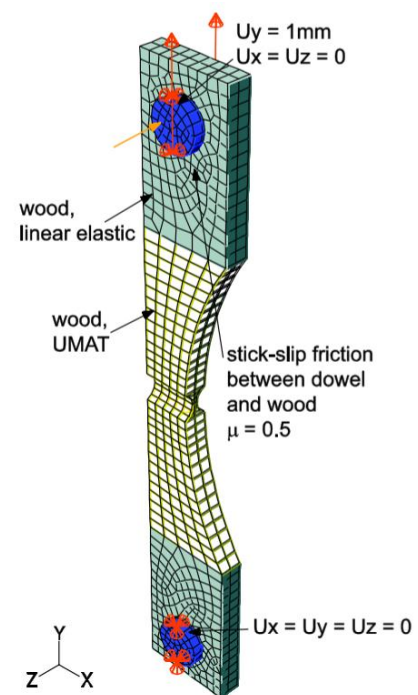


Figure 19: Dog-bone geometry. (38)

A simplification here used concerns the tensile and compressive strength which was here considered equal. Only one Hill failure surface was used for both plastic and brittle behaviour. Just like previously mentioned research, they also had convergence difficulties which they tackled in the same way. Additionally, parametric studies have been performed to

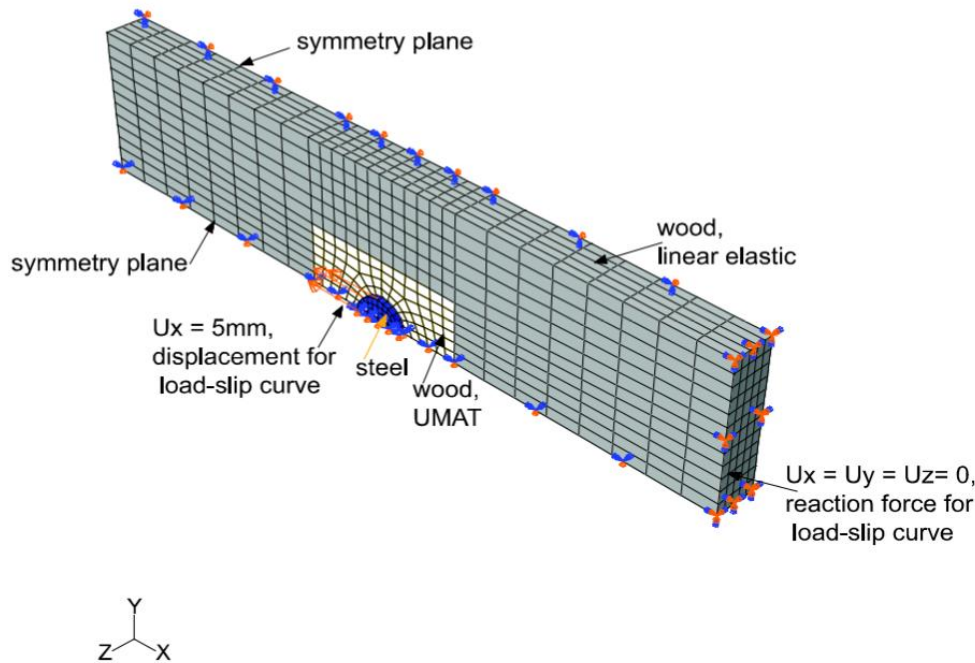


Figure 20: Embedment test. (38)

identify the effects of material parameters on the numerical results. 14 material properties were used to describe the behaviour of wood whereas only 4 were needed for the steel dowels: Young's modulus, Poisson coefficient, Yield strength and tensile strength. An optimisation was made regarding the activation of criteria to simulate the damage to wood. Element deletion rules were put in place and allowed to overcome extreme element distortion and mesh dependency. Convergence problems were found when elements were deleted in the contact region but solved by using a tolerance in the subroutine.

The model created managed to represent the failure initiation and propagation in the material, based on max stresses developed. Just like other researchers, a subroutine UMAT was written in FORTRAN and linked to Abaqus. To regularise the mesh, a crack band method was used when a localised failure was visible, meaning for the dog-bone geometry and not for the embedment tests. The authors have found that to correspond more closely to experimental results, it was sometimes necessary to vary the strength of wood or steel. This showcases how sensitive numerical models are to natural variations of material properties, numerical influences like the type of contact or even manufacturing accuracy. The authors also found that a refined mesh in the contact region was leading to an over estimation of the stiffness in that area and that a coarser mesh was preferable.

Conclusion to the short State of the art of numerical analysis of wood

In conclusion, there currently exists several ways to model wood varying in complexity as well as accuracy. From the microstructure to the macrostructure, the possibilities are numerous. One or several failure criteria can be chosen from an extensive existing list to try to catch as closely as possible the orthotropy and complexity of wood. At the moment, experiments are needed in order to compare with the numerical results as well as to calibrate the models. Research in the field of numerical modelling of wood is ongoing and essential as the possible outcome of accurately modelling wood would allow to perform abundant tests on a multitude of geometries in no time in comparison to experiments. Therefore allowing to optimise geometries, reducing the resources needed and increasing safety.

2.2. Introduction to the Finite Element Method and Abaqus

2.2.1. The Finite Element Method

Key aspects of the subsection: The Finite Element Method (FEM) is a numerical technique used to solve complex equations describing physical phenomena like structural analysis and heat transfer. Developed in the mid-20th century and popularized with the advent of computers, FEM discretizes structures into smaller elements (forming a mesh) for approximation. Despite its versatility and accuracy, FEM introduces modelling and discretization errors that can be mitigated through careful element selection and mesh refinement. However, challenges like rigid body motion, shear locking, and volumetric locking necessitate expertise in its mathematical principles and software tools.

The Finite Element Method (FEM) is a procedure allowing to numerically solve equations that would be too complex to handle by hand. Typically, it will aim to solve problems found in nature such as structural analysis, heat transfer, fluid dynamics and many more. Most of the time, they will be described with the use of equations expressed in differential or integral form. The first use of the term “Finite Elements” by R.W. Clough dates back to 1960 but the real start is in 1941 when Hrennikoff and Courant published papers about discretization’s similar to Finite Elements trying to solve particular Boundary Value Problems. The rise of computers in the 1960s and 1970s allowed this method to reach the success known today. (33) (39) Since then, FEM has now become essential to engineers and scientists analysing or designing any type of structure or complex part: from dams and buildings to aircrafts or electronic components.

Only a brief introduction of the Method and its key elements is here explained as books and courses are needed to get a complete understanding of this broad field. Therefore, the interested readers can reach for the following sources hereby mentioned. (8) (33) (36) (39)

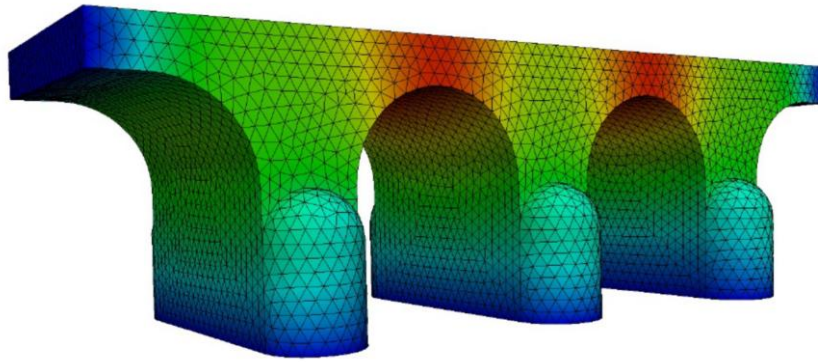


Figure 21: Result of a Finite Element Method simulation on a bridge. (39)

For the FEM to approximate the solutions to Partial Differential Equations (PDE) describing the behaviours in the nature, it converts them into their so-called weak form often using the principle of virtual work or variational methods. Among the many advantages of the weak form, it allows for less strict requirements on the continuity of the solution, a characteristic required by the numerical approximation.

The power of FEM is that it discretizes (“breaks down”) complex geometries of any size into smaller and simpler parts called Finite Elements. On these elements, approximate solutions can be computed. There exist many types of Finite Elements as visible on the non-exhaustive Figure 22. The choice will depend on the geometries to model, the dimension modelled (1D, 2D or 3D), the program used, the accuracy aimed for as well as the computational cost.

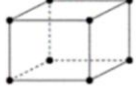
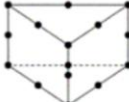
	1D	2D		3D		
	Beams	Triangles	Quadrilaterals	Tetrahedrons	Hexahedrons	Pentahedrons
FIRST ORDER	 2-noded	 3-noded	 4-noded	 4-noded	 8-noded	 6-noded
SECOND ORDER	 3-noded	 6-noded	 8-noded	 10-noded	 20-noded	 15-noded

Figure 22: Types of Finite Elements. (8)

Typically, 2D Quadrilaterals and 3D Hexahedrons have better convergence properties but have limits to model any type of geometries. Whereas Triangles and Tetrahedrons can model almost any type of geometries. Together, the elements form what is called a “mesh”. The behaviour of each Finite Element is depicted thanks to shape functions representing how the physical quantities (displacement, temperature, etc) vary. Afterwards, the approximations are assembled to form a global system of equations modelling the reaction of the structure.

Efficient mesh generators are key to a good numerical analysis as well as advanced visualisation techniques.

It is important to note that FEM only provides an approximation to the exact solution since the “exact” analytical value or variation is often highly complex and most of the time, unknown. The discretization of the geometry into Finite Elements also introduces another approximation. Therefore, when using FEM, two sources of error are present: the modelling error and discretization error. The first one can be reduced by using conceptual and structural models as efficient as possible. The second one can be reduced by using a finer mesh or higher order polynomial expansions. (33)

Every structure is three-dimensional (3D). However, in Finite Element Analysis, some can be approximated by 1D or 2D structural models therefore reducing the computational cost and time significantly. Notably, FEM permits to approximate the behaviour of a structure with an infinite number of Degrees of Freedom (DOFs) by a structure with a finite number of DOFs. The structure being linked to the external forces by the use of a system of algebraic equations expressing the equilibrium of the structure.

Two main categories of 2D analysis exist. (33) Plane stress problems and plane strain problems. The former contains structures for which one of its dimensions – thickness – is much smaller than the other two. The loads must also be contained in the middle plane of the structure. For example, deep beams, plates or buttress dams. On the other hand, plane strain problems correspond to structures for which one of its dimensions (length) is larger than the other two. The loads must be uniformly distributed along this dimension. For example, containing walls, pipes, tunnels and foundations, etc. Another possibility to simplify a 3D structure with FEM to reduce computational cost is the use of symmetries. This will be detailed in the following subsections.

The Finite Element Method involves the following steps for the analysis of a structure where the displacements of the nodes of the Finite Elements¹³ are sought after. (33)

1. Description of the geometry of the structure, its boundary conditions and the load. Selection of a structural model.
2. Definition of the material properties.
3. Definition of the scope of the analysis: small or large displacements, static or dynamic analysis.
4. Discretization of the structure into a mesh of finite elements.
5. Obtention of the stiffness matrix $\mathbf{K}^{(e)}$ and the load vector $\mathbf{f}^{(e)}$, for each element.

¹³ The extremities of the edges of Finite Elements are typically called nodes.

6. Assembly in a global stiffness matrix $\mathbf{K}$ and load vector $\mathbf{f}$ of the element stiffness and load terms.
7. Resolution of the global system of linear simultaneous equations $\mathbf{K}\mathbf{a} = \mathbf{f}$ for the unknown variables $\mathbf{a}$.
8. Evaluation of the strains and stresses derived from the displacements $\mathbf{a}$ and computation of the reactions at the restrained nodes.
9. Visualisation and interpretation of the results.
10. Verification of the suitability of the structural model, its finite elements, etc.

As depicted in the Figure 23, there are several hierarchical levels allowing to zoom in or out on the elements of the structure. Associated to these levels are reference systems to orient ourselves. Each hierarchical level has a dedicated generalized force as well as a generalized displacement, together linked by a constitutive law. In order to go from one generalized displacement to another one (of different hierarchical level), compatibility equations can be used. For generalized forces, equilibrium equations must be used. The sections deformations and forces are dependent on the Finite Element formulation. Whereas Euler-Bernoulli assumptions allow to determine the strains and stresses from the section deformations and section forces, respectively. (39)

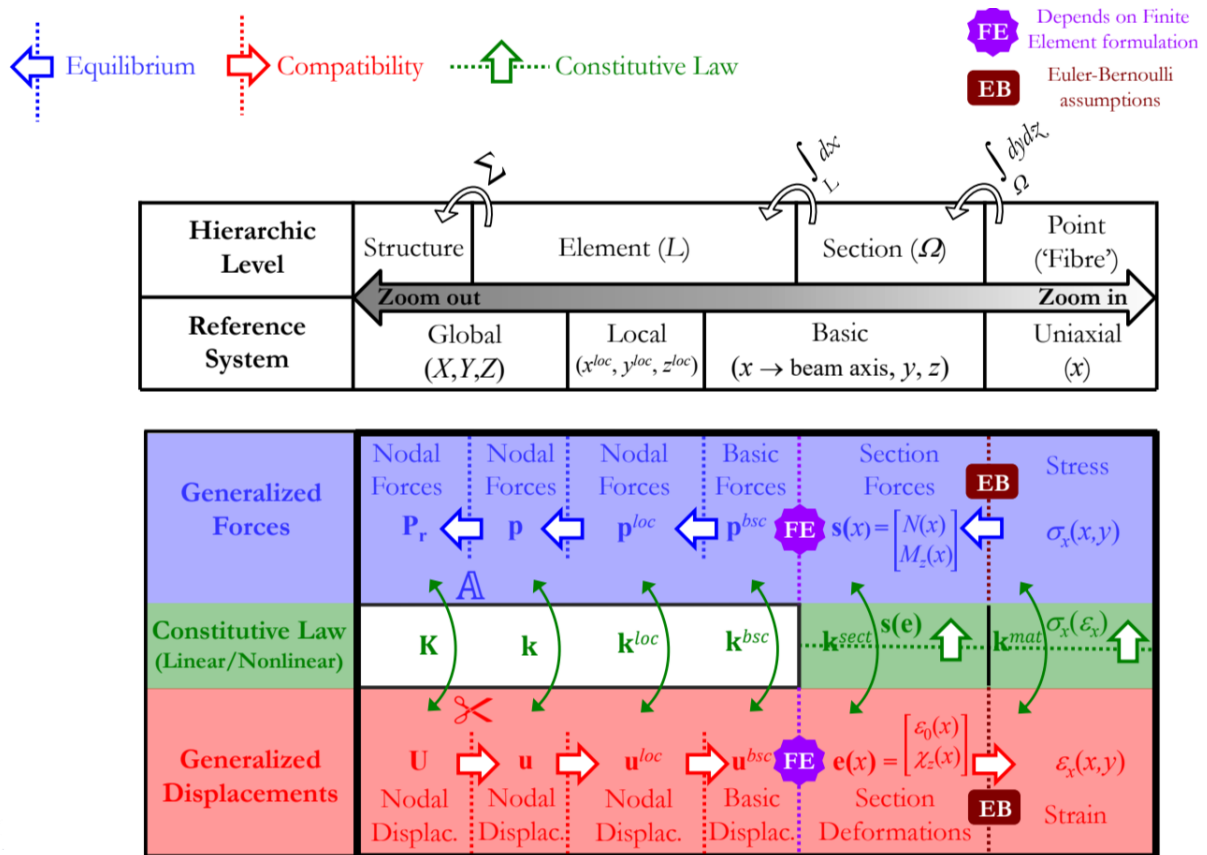


Figure 23: Summary of Numerical Analysis of a structure. (39)

When performing numerical analysis, issues can arise affecting the results. (39) Here are four common issues to be aware of to limit them.

First, **rigid body motion**, is linked to the definition of the boundary conditions. In fact, if the solid can move alone as a rigid body, then the rigidity matrix will not be invertible as one of the eigenvalues of the system is zero. Therefore, no energy will be required to activate this mode and most of the time, the software will simply not run. It is essential to ensure that all degrees of freedom are appropriately constrained.

A second common issue concerns **shear locking**. It is a phenomenon appearing for fully integrated linear elements subjected to bending. Precisely, shear stresses appear because the edges cannot bend therefore leading to a lower overall deformation and an over stiffness of the element. To avoid such problem, higher-order elements can be used or reduced integration techniques (often directly available in the software).

A third common issue in numerical analysis is **volumetric locking**. It corresponds to the over stiffening of elements when the material is constrained to deform volumetrically, usually when nearly incompressible (for Poisson's ratio ν close to 0.5 for example). This phenomenon appears for fully integrated elements such as 4-noded quadrilaterals or 8-noded hexahedrons. Selective or reduced integration can solve the problem.

Finally, the **hourglass** phenomenon (as seen in Figure 24) is a numerical instability easily recognizable by its zig zag shape on the deformed elements. This is due to non-physical zero-energy deformation modes occurring for under-integrated elements. It can be solved by using stabilization techniques or fully integrated elements.

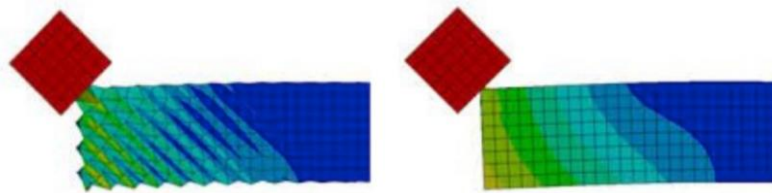


Figure 24: On the left, Hourglass phenomenon. On the right, the correct result. (39)

To conclude, FEM has limitations as it requires a deep understanding of the mathematical concepts behind, a software and expertise on this software to successfully implement the FEM. Approximation errors must be reduced as much as possible with a proper mesh and element selection. Computational resources must also be available. However, FEM has many advantages as it allows to model complex geometries and to change parameters in little time. It is also a highly versatile method allowing to model very diverse real-life problems with a high accuracy.

2.2.2. The commercial software ABAQUS

Abaqus is a commercial finite element analysis software developed by Dassault Systems Simulia. Abaqus offers three different modules optimal for different applications. (8) (36) (40)

- Abaqus CAE corresponds to the complete Abaqus environment and allows to model mechanical or structural components, perform finite element analysis' and to visualize the results.
- Abaqus Explicit is aimed for dynamic finite element analyses. It uses an explicit integration scheme.
- Abaqus Standard, on the contrary, uses an implicit integration scheme.

As depicted above, a solver can be either explicit or implicit. The difference between the two revolves around the time-stepping used for the numerical simulation. The explicit solver estimates the value of the variable at the next time step using known values at the current time step. On the other hand, an implicit solver calculates the solution at the next time step based on unknown values at the next step itself. The solver therefore solves a system of equations in which the current and next time steps are involved. Abaqus CAE is here used to model the structure and its parameters, and it uses Abaqus Standard to perform the analyses.

In order to understand the theory behind the software, the Abaqus Theory Guide (41) is a great source of information for the interested readers. To understand how the interface and how to practically use the software, the Abaqus CAE User's manual (40) and the practical exercises of the course "Numerical Analysis of Civil Engineering Structures" (39) are also extremely useful.

Abaqus does not contain units apart from angle measurements. Therefore, the units chosen by the user must be consistent. The units here considered are visible on the Table 9. In the Figure 25, the key steps of Abaqus are summarized. In fact, a complete analysis in Abaqus starts with a preprocessing step in which the structure is modelled using CAE. This step generates an input file used for the next step: the simulation. It can be performed either with Abaqus Standard or Explicit as mentioned above. The files generated by the simulation are called output files and they are used for the last main step: the postprocessing. This step is also key as it allows the visualization of the results through images or graphs: deformations, displacements, reaction forces, constraints, etc.

Dimension	SI units	Abaqus units
Length	m	mm
Force	N	N
Mass	kg	t
Time	s	s
Stress	Pa [N/m ²]	MPa [N/mm ²]
Energy	J	mJ
Density	kg/m ³	t/mm ³

Table 9: SI and consistent units chosen in Abaqus.

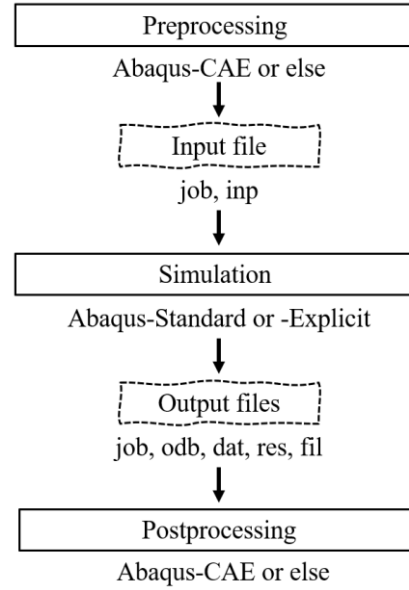


Figure 25: Key steps of Abaqus.

2.3. Material models

Key aspects of the section: Wood, as outlined in Section 1.5, exhibits properties that vary based on numerous factors, including direction, hygroscopicity, and temperature. Section 2.3 explains how wood can be modeled numerically, from a highly simplified model to an orthotropic model with circular coordinates. This provides readers interested in developing a numerical model of wood with a range of options, from simpler to more advanced approaches. The effect of the chosen material model on the results will be explored in Section 2.4.

In this section, four material models used in Abaqus are described (see Table 10). It was decided to use four different material models to compare the results. The material models differ in difficulty to implement them and therefore it is interesting to compare the results of a simple material model against more complex ones.

The first material model considered is an elastic isotropic model using a Young's modulus of 11'000 [MPa] and a Poisson coefficient of 0.3. This is a very simple model which does not precisely model wood and is usually intended for materials like steel. In fact, wood is an orthotropic material as discussed earlier. Therefore, this simplistic isotropic material model is performed here for comparison purposes against more complex material models and for its simplicity to implement as no material orientation of the fibres is to be determined. It is important to note that the rigidity coefficient is not asked by Abaqus for this model because it automatically computes it (see Equation 5). The rigidity coefficient in this case amount to 7'150 [MPa] which is approximately ten times more than what is typically considered for wood.

Material models	Description
n°1	Elastic – isotropic E = 11 000 MPa Poisson coefficient: 0.3
n°2	Elastic – isotropic for wood E = 11 000 MPa G = 690 MPa Poisson coefficient: 0.3
n°3	Rectangular coord., Mechanical > Elastic > Engineering constants Corresponds to an orthotropic model.
n°4	Cylindrical coord., Mechanical > Elastic > Engineering constants Corresponds to an orthotropic model.

Table 10: Material models considered in Abaqus.

$$G = \frac{E}{2 \cdot (1 + \nu)}$$

Equation 5: Rigidity coefficient for an isotropic material.

With:

- E: Young modulus.
- ν : Poisson coefficient.

Before detailing the material model number 2, the sections considered in Abaqus must be explained (see Figure 26). The first section corresponds to a simple squared element of timber assuming the annual rings going from bottom to top. This is a hypothesis as real-life squared element can have this configuration or a totally different annual ring configuration. The three orthotropic axes are depicted: Longitudinal (L), Radial (R) and Tangential (T). The second section considered approximates the geometry of a log which assumes a perfectly circular element with the annual rings as depicted in the Figure 26. Also with the three mutually orthogonal axes of wood. The steps to implement, in Abaqus, the fibre orientation of a wood element and its annual rings is depicted in the Annex 5.4.

Now the material model number 2 can be described. It considers an elastic material like the material model number 1 but considers that wood has a much lower rigidity modulus which does not follow the Equation 5. This material model is however much more complex to implement in Abaqus because it requires the user to determine the orientation of the fibres and the annual rings in the wood elements as described above. The Young modulus considered, and Poisson coefficient, is the same as for material model 1. The rigidity coefficient however is

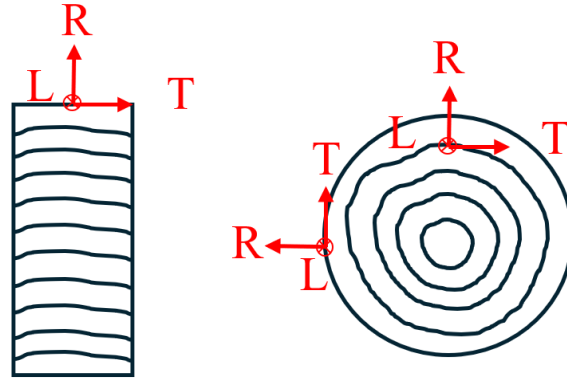


Figure 26: The sections considered in Abaqus.

taken as 690 [MPa] to correspond to a Douglas C24 element. The three parameters are here the same in the three axes of wood.

The material number 3 corresponds to an orthotropic material model for wood for squared element as depicted in the Figure 26. As the timber element is a parallelepiped rectangle, rectangular coordinates are chosen. As Douglas has been chosen as the material considered for the experimental campaign (see Chapter 3), it was decided to also consider Douglas for this part of the Master Thesis.

The parameters considered are depicted in the Table 11. The first three parameters: E_L , E_R , E_T , correspond to the Young's moduli of the material in the longitudinal, radial and tangential directions, respectively. G_{LR} , G_{LT} , G_{RT} correspond to the rigidity coefficients (also called "Shear moduli") in the longitudinal-radial plane, longitudinal-tangential plane and radial-tangential plane, respectively. Lastly, the μ coefficients represent the Poisson ratios in the six planes of interest, also denominated by two letters. They were explained in more details in Section 1.5. The twelve parameters can be obtained experimentally as described in the Annex 5.2.

E_L [MPa]	E_R [MPa]	E_T [MPa]	μ_{LR}	μ_{LT}	μ_{RT}
11 000	770	570	0.292	0.449	0.39
G_{LR} [MPa]	G_{LT} [MPa]	G_{RT} [MPa]	μ_{TR}	μ_{RL}	μ_{TL}
720	880	80	0.374	0.036	0.029

Table 11: Material parameters considered for the material models 3 and 4. (30)

Finally, the material model number 4 also considers an orthotropic material model but it uses cylindrical coordinates and is therefore dedicated to use for circular sections like the one described in Figure 26. The material parameters are the same as for the material model number 3.

2.4. Introductory exercises of wooden beams

Key aspects of the section: This section presents two exercises, a simply supported beam and an overhanging beam, to evaluate how the choice of material model affects the simulation results for wood. The exercises are performed for a square section as well as a circular section. The simplest model, isotropic, consistently exhibits the greatest deviation from analytical results, while the isotropic model tailored for wood and the orthotropic model show only minor differences compared to the analytical solution.

In this section, two main introductory exercises are studied. These exercises were chosen to better learn how to model the material wood and compare results depending on the decisions taken regarding the material model, the section considered, the application of the load, the finite elements to use as well as their density. This is in fact a preparation for the more complex numerical simulations to be performed on RoboTronc geometries.

2.4.1. Exercise one: simply supported beam

The **first exercise** concerns a simple wooden beam resting on two supports: a pinned support and a roller support, as visible in the Figure 27. The load here considered is a static linear load spread equally all over the top surface of the beam, simulating the loads of the floor of a building. Two sections are considered, a rectangular one simulating squared timber elements and a circular one simulating logs like the ones considered for RoboTronc.

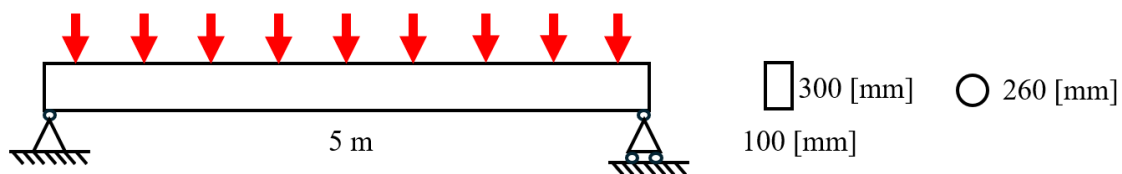


Figure 27: Introductory exercise 1.

For each section (squared and circular), few material models will be performed and compared to analytical results like the ones performed in an engineering study following EC5 (2). Finite elements suitable for the situation will be chosen and a mesh convergence study will be performed.

First, the case of the **squared section** and its **analytical description** are explained. The Figure 28 till Figure 30 highlight the placement of the steel boundary conditions and the dimensions of the cross-section.



Figure 28: Wood beam (brown) and steel support (grey).

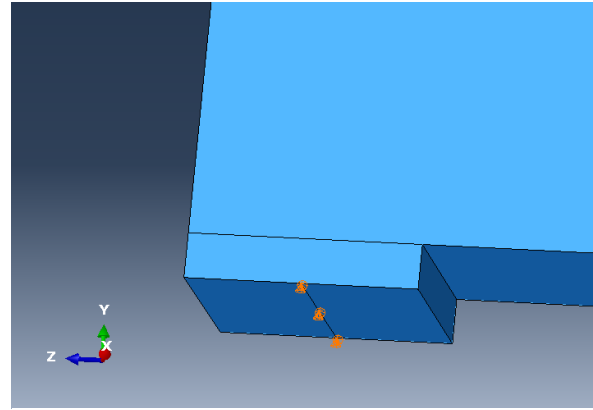


Figure 29: Boundary conditions in Abaqus.

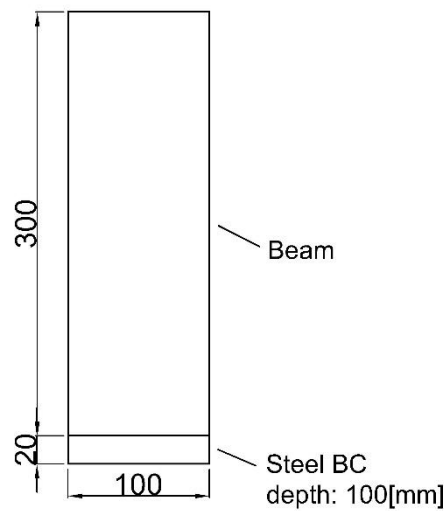


Figure 30: cross-section of the rectangular wood beam with its steel boundary condition.

Key parameters of the beam:

- Length of the beam: $L = 5 \text{ [m]}^{14}$
- Area of the section: $A = 0.3 \cdot 0.1 = 0.03 \text{ [m}^2\text{]}$
- Inertia of the beam: $I = \frac{b h^3}{12} = 2.25 \cdot 10^{-4} \text{ [m}^4\text{]}$
- Material : Douglas, C24

Load considered:

- Linear load: $q = 3.6 \text{ [kN/m]}^{15}$

¹⁴ The length of the beam is considered from one extreme face to its opposite extreme face, not from support to support. This may induce a difference of results between the numerical models and the analytical results.

¹⁵ The dead weight of the beam was neglected for the exercise 1 and 2 as it could be considered as included in the linear load applied.

Internal efforts:

- Maximum bending moment: $M_{max} = \frac{q L^2}{8} = 11.25 [kNm]$
- Maximum shear stress: $V_{max} = \frac{q L}{2} = 9 [kN]$

Deflection of the centre of the beam:

- Deflection due to the bending moment: $\delta_M = \frac{5 q L^4}{384 E I} = 11.84 [mm]$
- Deflection due to the shear: $\delta_V = \frac{3 q L^2}{20 G b h} = 0.65 [mm]$
- Total deflection at centre of the beam: $\delta_{Tot} = \delta_M + \delta_V = 12.49 [mm]$

Stresses due to bending, at the centre of the beam.

- $\sigma_m = \frac{M_{max}}{W_{el}} = \frac{q L^2}{8} \cdot \frac{1}{\frac{b h^2}{6}} = \pm 7.5 [MPa]$

With W_{el} the ‘Elastic section modulus’.

With compression on the top fibre and tension in the bottom fibre.

Back to Abaqus and the modelling of this beam using the material models previously stated. The exact steps to follow in Abaqus are described in the Annex 5.4. The results of the tests performed on Abaqus are visible in Table 12.

The first model used is the isotropic model for wood (described section 2.3). For this test, the boundary conditions were applied using an intermediary steel element as boundary conditions as visible in Figure 28. The support consists in a steel element of $100[mm] \cdot 100[mm] \cdot 20 [mm]$. The contact between the two elements is considered as ‘rough’ as there is no slippage wanted between the two surfaces. And the boundary conditions, a pinned support and a roller support, are applied to the bottom of the steel element, on a centred perpendicular line as visible in Figure 29. This support aims at representing the closest possible the support considered analytically.

Model	Geometry	Deflection [mm]	Difference [%]	Max stress [MPa]
Analytical model	Squared	12.49	[-]	7.50
Isotropic model for wood	Squared	12.52	+0.24	7.54
Isotropic model	Squared	11.92	-4.56	7.47
Orthotropic model	Squared	12.34	-1.2	7.53
Analytical model	Circular	12.64	[-]	6.49
Isotropic model for wood	Circular	12.53	-0.87	6.49
Isotropic model	Circular	12.20	-3.48	6.35
Orthotropic model	Circular	12.59	-0.40	6.53

Table 12: Results obtained for the exercise 1.

The specifications of the exercise, including the load, were inspired from an exercise of the course of “Structure en bois” (17). The situation represents a realistic floor beam. In Abaqus, the load has been taken as a uniformly applied pressure load on the top surface of the beam. Regarding the mesh, different finite elements have been compared and a mesh convergence study has been performed. The details are visible in the Annex 5.3. The outcome of this study shows that a linear tetrahedral finite element is highly performant with an approximate global size of the seeds¹⁶ of 20, 15 or 10. A global seeding size of 10 [mm] was chosen here for the introductory exercises.

As visible in the Table 12, the deflection of the beam at half-span is 12.52 [mm] when using this material model (“isotropic for wood”) as opposed to 12.49 [mm] for the analytical results. This is a difference of only +0.24 [%]. The distribution of the stresses¹⁷ along the beam is visible in the Figure 31 and in the Figure 32. A deformation scale factor of 20 is used to highlight the shape of the deflection of the beam. This Factor corresponds to a multiplier applied to the results of the displacements obtained to better visualize the different phenomena. For example, a deformation scale factor of 1 shows the exact displacements calculated by the software whereas a deformation scale factor of 20 takes the same displacements and multiply them by 20.

¹⁶ The “approximate global size of the seeds” is described in detail in the Annex 5.3.

¹⁷ In Abaqus, the stresses are denoted by “S”. The digits allow to identify the direction of the stresses considered. For example, S11 denotes here the stresses in the longitudinal axis (x-axis) because S11 corresponds to the normal stress acting on the plane perpendicular to the 1-axis (x-axis), in the direction of the 1-axis. S22 and S33 are also normal stresses whereas the other stresses (S12, S13, 23) denote shear stresses.

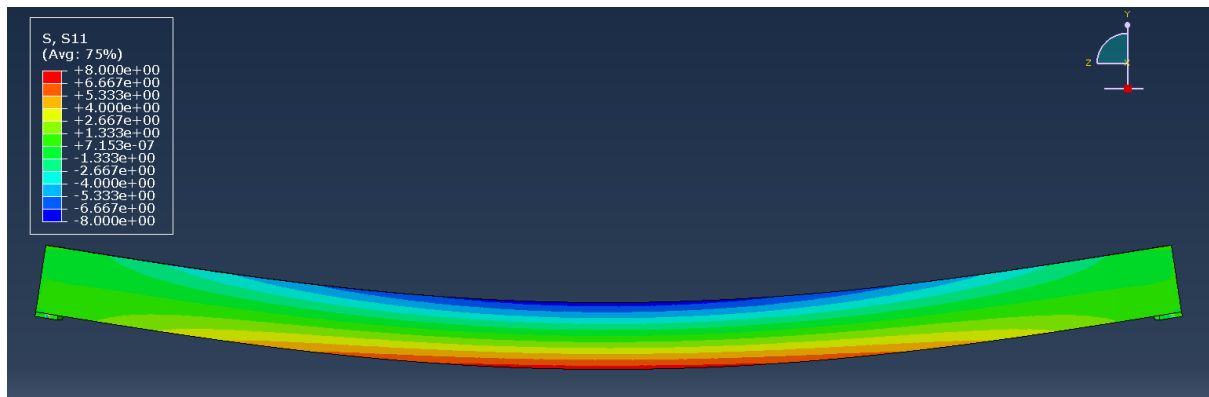


Figure 31: Lateral view of the stresses in the squared beam when considering an Isotropic for wood material model. Deformation scale factor: 20.



Figure 32: Top view of the stresses in the squared beam when considering an isotropic for wood material model. Deformation scale factor: 20.

The same test is carried out but using the very basic isotropic material model. The results showcase a deflection of 11.92 [mm] at mid-span which represents a difference of -4.56 [%] with the analytical results. This difference may initially appear significant compared to the other results but in fact, it is relatively minor when considering the material variability of wood in between specimens of a same species as well as the influence of other parameters such as a proper boundary condition application.

Lastly, for the squared section, the orthotropic material model has been performed and showcases a deflection at mid-span of 12.34 [mm], representing a difference of -1.2 [%]. This difference is not very high, and it is important to remember that the orthotropic material model characterizes the material differently than the analytical calculations. Therefore, this test situation does not showcase that the orthotropic material model is less precise than the isotropic model for wood. They are simply different in their hypothesis, and this test simply shows the difference in results between the two models.

A pattern can now be observed when comparing the three material models used in Abaqus for this simple exercise. In truth, the isotropic model for wood is here the closest to the analytical results which makes sense as it is also the material model with the closest hypothesis'

used compared to the hypothesis' considered analytically. The second closest is the orthotropic model and the last one is the simple isotropic material model.

Now, the same tests are carried out on the **circular section**. Once again, the description starts with the **analytical results** to be compared to the numerical results afterwards. The diameter of the circular section is chosen to be the same as the rectangular section, before the top and bottom cuts on the circular section are performed. The impact of these cuts on the results will be discussed in this section. A 3D perspective of the cuts on the top and bottom of the circular beam is visible in the Figure 33 and Figure 34 as well as the boundary conditions. The exact dimensions of the cross-section together with its steel boundary condition is visible in the Figure 35.

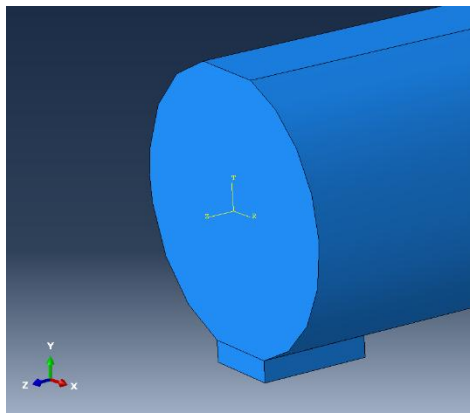


Figure 33: Top cut and bottom support.

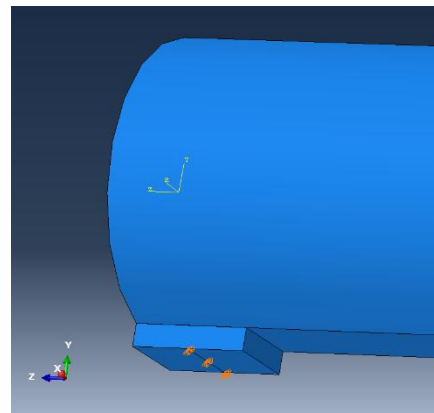


Figure 34: Boundary conditions in Abaqus.

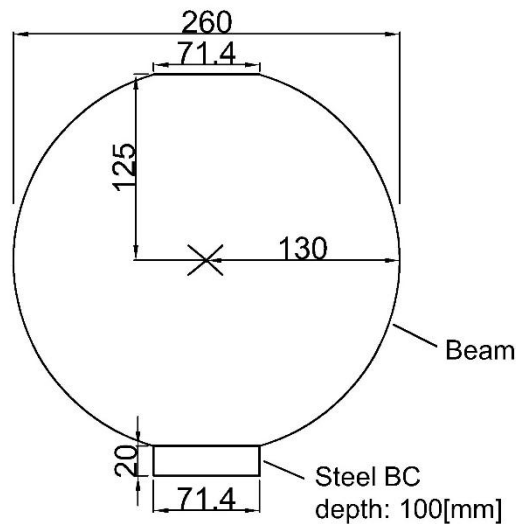


Figure 35: cross-section of the circular wood beam with its steel boundary condition.

Key parameters of the beam:

- Length of the beam: $L = 5 \text{ [m]}$
- Gross area of the section: $A_{\text{gross}} = \pi r^2 = 0.053 \text{ [m}^2\text{]}$
- Net Area of the section¹⁸: $A_{\text{net}} = 0.053 \text{ [m}^2\text{]} - 0.000476 \text{ [m}^2\text{]} = 0.052524 \text{ [m}^2\text{]}$
- Gross inertia of the beam: $I_{\text{gross}} = \frac{\pi d^4}{64} = 2.25 \cdot 10^{-4} \text{ [m}^4\text{]}$
- Net inertia of the beam: $I_{\text{net}} = 2.1655 \cdot 10^{-4} \text{ [m}^4\text{]}$
- Material : Douglas, C24

Load considered:

- Linear load: $q = 3.6 \text{ [kN/m]}$

Internal efforts:

- Maximum bending moment: $M_{\text{max}} = \frac{q L^2}{8} = 11.25 \text{ [kNm]}$
- Maximum shear stress: $V_{\text{max}} = \frac{q L}{2} = 9 \text{ [kN]}$

Deflection of the centre of the beam:

- Deflection due to the bending moment: $\delta_M = \frac{5 q L^4}{384 E I_{\text{net}}} = 12.30 \text{ [mm]}$
- Deflection due to the shear: $\delta_V = \frac{q L^2}{8 G A_v} = \frac{q L^2}{8 G 0.9 A_{\text{net}}} = 0.34 \text{ [mm]}$
- Total deflection at centre of the beam: $\delta_{\text{Tot}} = \delta_M + \delta_V = 12.64 \text{ [mm]}$

Stresses due to bending, at the centre of the beam.

- $\sigma_m = \frac{M_{\text{max}}}{W_{el}} = \pm 6.49 \text{ [MPa]}$

With W_{el} the ‘Elastic section modulus’.

With compression on the top fibre and tension in the bottom fibre.

It is important to note that the circular beam requires a horizontal cut on its top surface in order to properly apply the load as visible in Figure 33. In the same Figure, the bottom support can also be observed. As the beam is originally circular, a horizontal bottom cut is also required to connect the steel support horizontally as visible in Figure 34 together with the

¹⁸ As visible in the figure , the circular beam element needs to be made horizontal for the application of the load and for the application of the boundary conditions. This affects the cross-sectional area as well as the inertia of the beam.

boundary conditions used in Abaqus. The analytical calculations were also carried out without considering the cuts made in the element on the top and down of the beam and the difference showcases:

- A reduction of 0.9 [%] of cross-sectional area
- A reduction of 3.76 [%] of inertia
- An increase in the deflection of 3.67 [%].

Therefore, what appeared to be a small cut on the top and down of the beam to have a horizontal surface to apply the loads and boundary conditions has a rather significant impact on the results. The cross-sectional area has a low reduction, but the material removed from the beam is placed in a critical location for the inertia of the beam. This highlights the negative impact of small cuts made at the top and down extremities of the beam.

Back to Abaqus and the modelling of this beam using the material models previously stated. The steps to follow in Abaqus are described in the Annex 5.4. The results of the tests performed on Abaqus are visible in Table 12.

The first model to compare with the analytical results is the isotropic model for wood. The deflection observed at mid-span in Abaqus is 12.53 [mm], a reduction of 0.87 [%] compared to the analytical results. The distribution of the stresses is visible in the Figure 36 and in the Figure 37. Afterwards, the basic isotropic material model is also carried out and highlights a deflection of 12.20 [mm]. This amounts to a difference of -3.48 [%] with the analytical results. Lastly, the orthotropic model showcases a deflection of 12.59 [mm] meaning a difference of -0.40 [%] with analytical results.

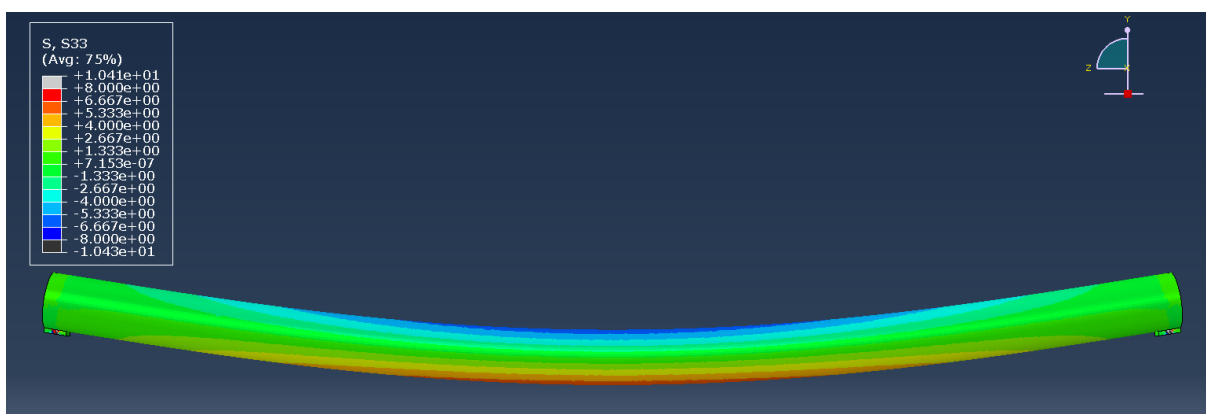


Figure 36: Lateral view of the stresses in the circular beam when considering an isotropic for wood material model. Deformation scale factor: 20.

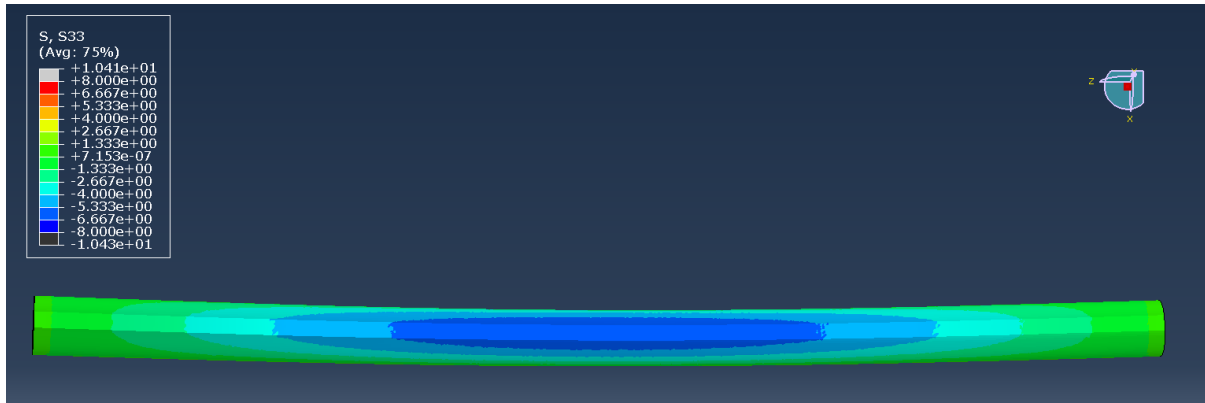


Figure 37: Top view of the stresses in the circular beam when considering an isotropic for wood material model. Deformation scale factor: 20.

The following pattern (from Table 12) can be observed for the squared section. The isotropic model for wood is the closest model to the analytical results whereas the orthotropic model comes second. The last one is the simpler isotropic material model. Regarding the circular section, a slightly different pattern can be observed. The closest is now the orthotropic mode closely followed by the isotropic model for wood. The last one is still the isotropic model. Regarding the stresses, they are generally close to the calculated analytical stresses. Out of the six tests here performed in Abaqus, five of them have a lower deflection than what was anticipated analytically. Therefore, the analytical calculations of the deflection are here mostly safe-sided.

Globally, the circular section showcases a slightly higher deflection than its rectangular counterpart with 1.19 [%] of difference for the analytical calculations. This is mainly due to the reduction of its inertia (necessary to apply the loads and for the supports), increasing the deflection by 3.67 [%]. However, the circular section resists better to shear due to a better A_v coefficient of 0.9 for the circular section instead of 0.83 for the rectangular section and to a higher cross-sectional area. Therefore, a total difference of 1.19 [%] is calculated between the rectangular and circular sections.

2.4.2. Exercise two: overhanging beam

The **second exercise** concerns an overhanging beam as depicted in the Figure 38. The beam is the same length as previously and the same two sections are considered. Just like exercise 1, different material models are tested to compare the accuracy of each of them. The results are summarized in the Table 13.

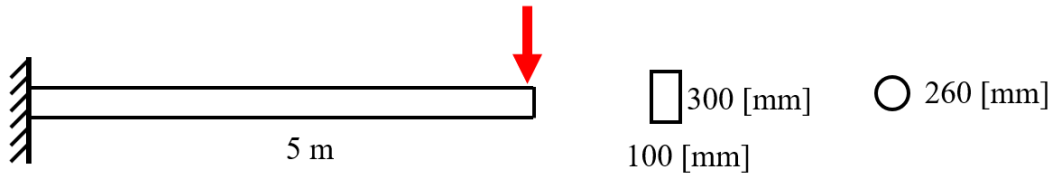


Figure 38: Introductory exercise 2.

Like for the exercise 1, the **squared section** and its **analytical description** are here explained.

Key parameters of the beam:

- Length of the beam: $L = 5 \text{ [m]}$
- Area of the section: $A = 0.3 \cdot 0.1 = 0.03 \text{ [m}^2\text{]}$
- Inertia of the beam: $I = \frac{b h^3}{12} = 2.25 \cdot 10^{-4} \text{ [m}^4\text{]}$
- Material : Douglas, C24

Load considered:

- Point load: $Q = 2.5 \text{ [kN]}$

Internal efforts:

- Maximum bending moment: $M_{max} = Q L = 12.5 \text{ [kNm]}$
- Maximum shear stress: $V_{max} = Q = 2.5 \text{ [kN]}$

Deflection of the centre of the beam:

- Deflection due to the bending moment: $\delta_M = \frac{Q L^3}{3 E I} = 42.09 \text{ [mm]}$
- Deflection due to the shear: $\delta_V = \frac{V L}{5/6 G b h} = 0.72 \text{ [mm]}$
- Total deflection at centre of the beam: $\delta_{Tot} = \delta_M + \delta_V = 42.81 \text{ [mm]}$

Stresses due to bending, at the centre of the beam.

- $\sigma_m = \frac{M_{max}}{W_{el}} = Q L \cdot \frac{1}{\frac{b h^2}{6}} = \pm 8.33 \text{ [MPa]}$

With W_{el} the 'Elastic section modulus'.

With tension on the top fibre and compression in the bottom fibre.

Model	Geometry	Deflection [mm]	Difference [%]	Max stress [MPa]
Analytical model	Squared	42.81	[-]	8.33
Isotropic model for wood	Squared	42.70	-0.26	8.88
Isotropic model	Squared	41.99	-1.92	8.19
Orthotropic model	Squared	42.92	+0.26	8.93
Analytical model	Circular	42.47	[-]	7.24
Isotropic model for wood	Circular	42.31	-0.38	7.5
Isotropic model	Circular	41.79	-1.60	7.4
Orthotropic model	Circular	42.58	+0.26	7.74

Table 13: Results obtained for the exercise 2

Back to Abaqus and first with the isotropic model for wood. The boundary condition is here easier and more straightforward in Abaqus as all the left extreme face of the beam is considered clamped. For the application of the load, it was decided to apply it to the centre of the opposite face and to force the points of this face to deform vertically the same way as the middle point (“Reference Point” in Abaqus with a “Tie” constraint).

The deflection calculated is the deflection at the right extremity of the beam. The stress displayed in the table is the stress visible in Abaqus on the top fibre. As this stress is next to the boundary condition, caution is necessary with this value because the results of the equations performed by Abaqus can be incorrect when close to the boundary conditions or point loads. The deflection obtained is 42.70 [mm] as opposed to 42.81 [mm] for the analytical model. This represents a difference of -0.26% which can be judged as very small. The measured stress of 8.88 [MPa] as opposed to 8.33 [MPa] represents a more significant difference of 6.19 [%]. As discussed above, this is probably due to the validity of the equations close to the boundary condition. The distribution of the stresses along the beam is visible in the Figure 39 to Figure 40.

Next, the isotropic model showcases a greater difference in the deflection of the beam with a difference with analytical results of -1.92 [%]. This is due to the inaccuracy of the material model to represent well the behaviour of wood in its three main axes.

Lastly, for the squared elements, the orthotropic model is tested. The difference between this model and the analytical model is of +0.26 [%] like for the isotropic model for wood. But this time, the orthotropic model shows a higher deflection than the analytical one.

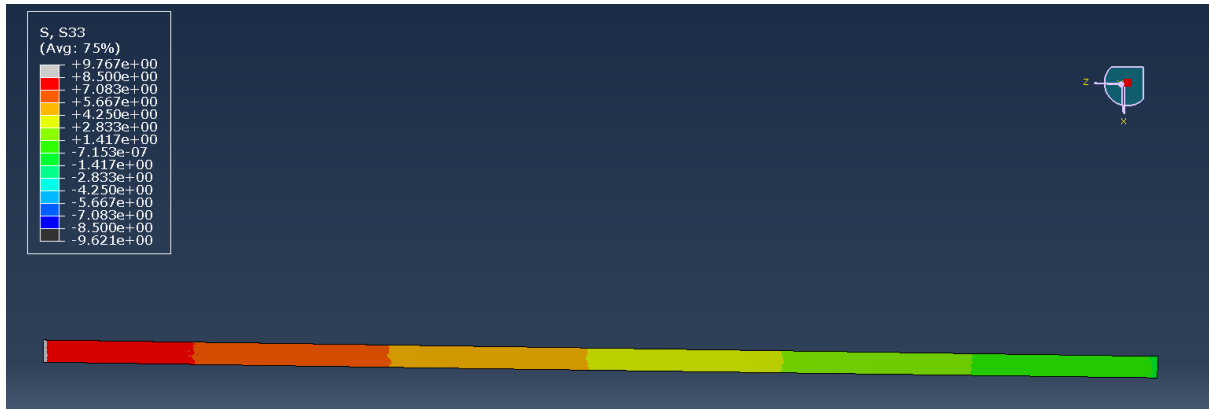


Figure 39: Top view of the stresses in the squared beam when considering an isotropic for wood material model.
Deformation scale factor: 20.

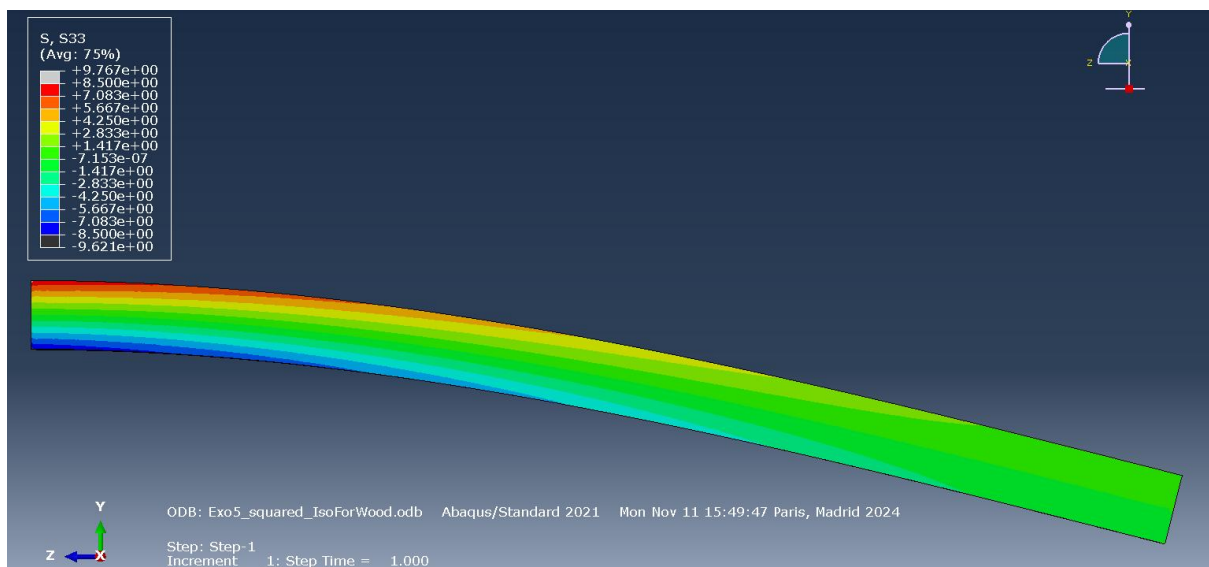


Figure 40: Lateral view of the stresses in the squared beam when considering an isotropic for wood material model.
Deformation scale factor: 20.

Now, the same tests are carried out on the **circular section**. Once again, the description starts with the **analytical results** to be compared to the numerical results afterwards.

Key parameters of the beam:

- Length of the beam: $L = 5 \text{ [m]}$
- Diameter of the beam: $d = 260 \text{ [mm]}$
- Area of the section: $A = \pi r^2 = 0.053 \text{ [m}^2\text{]}$
- Inertia of the beam: $I = \frac{\pi d^4}{64} = 2.25 \cdot 10^{-4} \text{ [m}^4\text{]}$
- Material : Douglas, C24

Load considered:

- Point load: $Q = 2.5 \text{ [kN]}$

Internal efforts:

- Maximum bending moment: $M_{max} = Q L = 12.5 \text{ [kNm]}$
- Maximum shear stress: $V_{max} = Q = 2.5 \text{ [kN]}$

Deflection of the centre of the beam:

- Deflection due to the bending moment: $\delta_M = \frac{Q L^3}{3 E I} = 42.09 \text{ [mm]}$
- Deflection due to the shear: $\delta_V = \frac{V L}{G A_v} = \frac{V L}{G 0.9 A} = 0.38 \text{ [mm]}$
- Total deflection at centre of the beam: $\delta_{Tot} = \delta_M + \delta_V = 42.47 \text{ [mm]}$

Stresses due to bending, at the centre of the beam.

$$\sigma_m = \frac{M_{max}}{W_{el}} = Q L \cdot \frac{1}{\frac{\pi D^3}{32}} = \pm 7.24 \text{ [MPa]}$$

With W_{el} the ‘Elastic section modulus’.

Back to Abaqus where the results can be observed in the Table 13. A similar pattern as before can be observed. The isotropic model for wood showcases a slightly lower deflection than the analytical model with a difference of -0.38 [%]. The distribution of the stresses in the beam is visible in the Figure 41 to Figure 42. The simple isotropic model showcases an even lower deflection with a difference of -1.60 [%] to the analytical deflection. Whereas the orthotropic model simulation displays a slightly higher deflection with 0.26 [%] of difference to the analytical model.

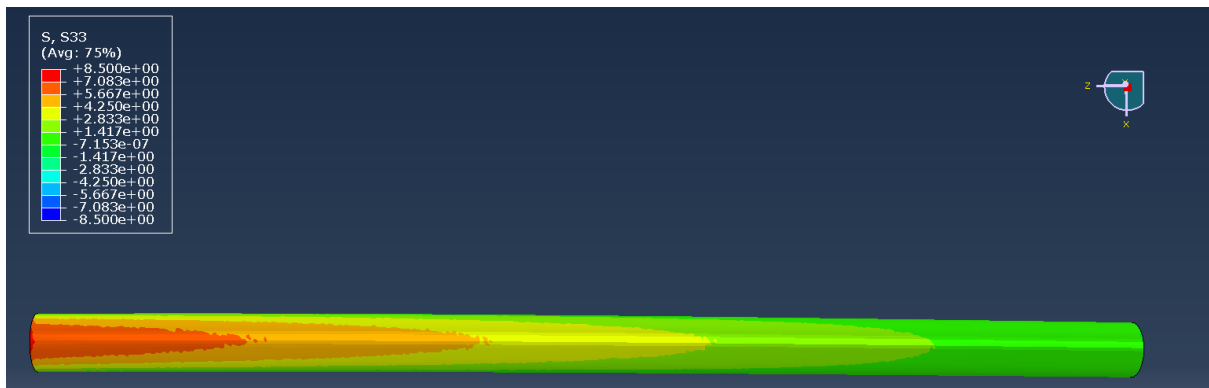


Figure 41: Top view of the stresses in the circular beam when considering an isotropic for wood material model. Deformation scale factor: 20.

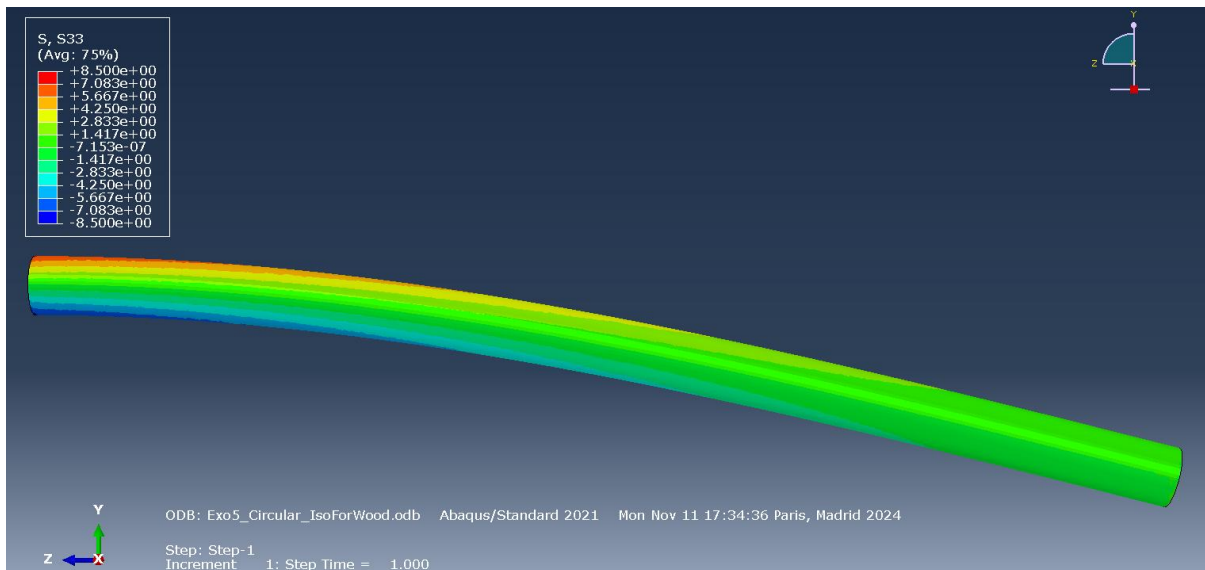


Figure 42: Lateral view of the stresses in the circular beam when considering an isotropic for wood material model.
Deformation scale factor: 20.

In conclusion, the exercises 1 and 2 highlight that the material model “isotropic for wood” bring results (in terms of deflection) close to the analytical calculations typically performed in an engineering study. This is due to very similar hypotheses between this material model and the way wood is modelled analytically. The orthotropic material model is also close to the analytical results whereas the isotropic material model is repeatedly the furthest with an underestimated deflection. Regarding the extraction of the stresses, the exercise 1 (simply supported beam) allowed relatively accurate data from the numerical simulations whereas the exercise 2 (overhanging beam) shows a wider range of results due to the proximity of the location of the maximum stresses to the boundary condition. The user must always keep in mind the limited validity of the equations when close to the boundary conditions.

2.5. Key parameters of numerical simulations of wood

This section aims at explaining three important parameters in a numerical model: the contact rules between the interfaces, the possible tolerances in between interfaces and the non-linearity of the material. In fact, they are essential as their impact on the results is significant. Here, the different possibilities are analysed and explained with simple examples allowing for quick comparisons.

To illustrate the possible contact rules, tolerances and non-linearities, a new exercise was chosen as described in Figure 43. This third exercise consists in a 5-m long beam clamped at its left extremities and partially resting at its right extremities on a 1-m tall column clamped at its bottom. The load applied at the top of the beam is the same as for the first exercise and corresponds to a linear load of 3.6 [kN/m] . The section of the beam and column is the same and corresponds to a rectangle of 300 [mm] by 100 [mm] . For this third exercise, the

orthotropic material model was chosen because it is the material model with the closest assumptions to the real behaviour of wood.

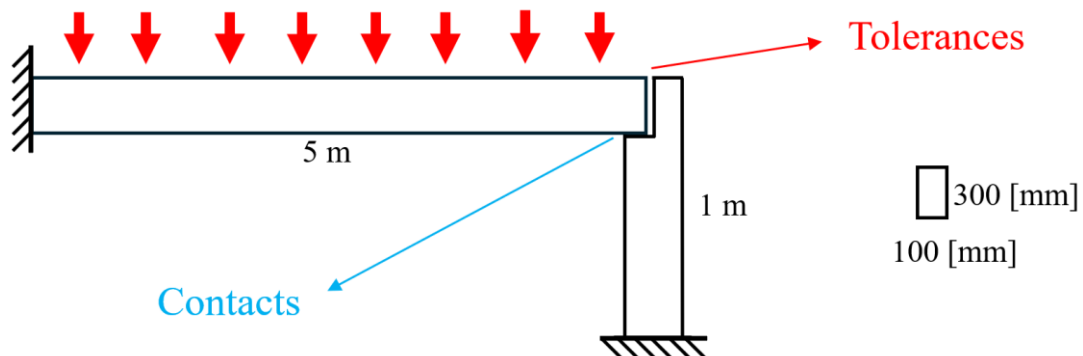


Figure 43: Scheme of the exercise 3.

2.5.1. Contacts

Key aspects of the subsection: This subsection examines various contact rules between a beam and a column, highlighting and discussing their impact on the results. The most suitable contact rule for modelling wood is then selected.

In Abaqus, there exists two main ways to specify the contact interactions between two surfaces. The first one “General contact” allows Abaqus to assess that there is a contact for any surface of a part touching another surface. This can be very useful when a contact between surfaces only appears during a simulation. The downside is that all contacts have the same contact interaction rule. The second main way of modelling contacts is through “Contact pairs”. This corresponds to specifying a contact interaction rule for a chosen surface with one another. Here, the “General contact” option was chosen for simplicity and because contact appears here only on two surfaces.

There exists a high number of contact rules in Abaqus. Here, only the most pertinent ones for wood are detailed and compared as visible in the Table 14. The tangential behaviour is to describe the interaction between surfaces in their tangential plane therefore focusing on the frictional effects and possible energy dissipations. On the other hand, the normal behaviour contact rule focuses on behaviour of the surfaces in their normal direction to the contact. Therefore, it describes the surface separation or compression. The cohesive contact rule can model the bond between surfaces for example when glued together. It can also evolve with the simulation (weakening or failing).

Contact rule	Contact sub-rule	Description
Tangential	Frictionless ¹⁹	The two surfaces can slide freely.
	Penalty	Allows to implement a friction coefficient.
	Static-kinetic exponential decay	Models the transition from static friction to kinetic friction.
	Rough	Imposes a constraint preventing the relative tangential movement between the surfaces.
Normal	“Hard contact” ²⁰	Impeaches the penetration but possibility of surface separation.
	Linear	Allows penetration between surfaces depending on a contact stiffness parameter.
Cohesive		Allows to simulate the interaction between surfaces with an initial adhesion that can weaken or fail.

Table 14: Main contact rules available in Abaqus.

There also exists two interesting contact rules that could be pertinent for wood: damage & fracture criterion. They were not studied in this master’s thesis as they require a mode advanced knowledge in numerical modelling than possible here. The first one, damage, can model the initiation and evolution of damage at the selected interface. The second one, fracture criterion, can model the final separation, or fracture, of the surfaces once the damage has evolved above a critical threshold. A phenomenon common when wood is loaded past its elastic properties.

It is important to note that the different contact rules can be used together. For example, it is frequent to use a Tangential behaviour together with a normal behaviour contact rule in order to model the behaviour of the surfaces in the two main directions of the contact.

¹⁹ If “General Contact” is selected but not a precise contact rule, then Tangential-frictionless is chosen by default by the software.

²⁰ If “General Contact” is selected but not a precise contact rule, then Normal-hard contact is chosen by default by the software.

The choice of contact rule in a model is essential as it influences the results like the deflections, strains or stresses in the elements. The Table 15 summarises the tests performed with different contact rules to identify the possible consequences on the deflection of the beam at its mid-span, the maximum stress in the beam as well as the horizontal displacement of the top of the column. The percentages in parenthesis have for reference the test 5. as it was decided to be the most pertinent contact rule for this situation.

The choice of contact rules will depend on the type of wood as well as the surfaces being put in contact: parallel to the grain of a squared timber element, parallel to the grain of a debarked log, perpendicular to the grain, etc. A good choice of contact rule and contact parameters (ideally from experiments) reduces numerical uncertainties. The mesh used corresponds to tetrahedral elements with seeds of an approximate global size of 20 for the beam as well as for the column. For this subsection about contacts, the tolerance between the right end of the beam and the column is set to 50 [mm] as visible in the Figure 44. This is to ensure that no contact happens in this area and only the contact on the bottom surface of the beam and top surface of the column happens.

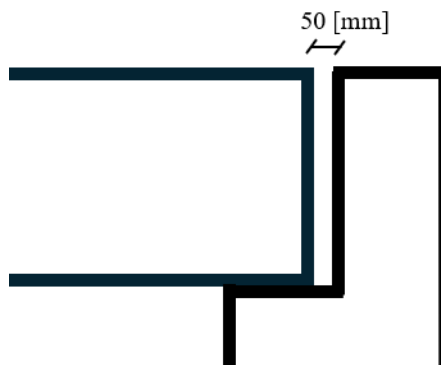


Figure 44: Tolerance of 50 [mm] between the beam and the column for the subsection “Contacts”.

Test	Contact rule	Contact sub-rule	Maximum vertical displacement of the beam [mm]	Maximum stress in the beam [MPa] ²¹	Maximum horizontal displacement of the column [mm] ²²
1.	Tangential	Frictionless	5.32 (3.10 [%])	4.09 (0.74 [%])	0.16
	Normal	“Hard contact”			
2.	Tangential	Rough	5.01 (-2.91 [%])	4.03 (-0.74 [%])	-0.37
	Normal	“Hard contact”			
3.	Tangential	Frictionless	5.33 (3.29 [%])	4.09 (0.74 [%])	0.16
	Normal	Linear			
4	Cohesive		4.91 (-4.84 [%])	3.96 (-2.46 [%])	-0.40
	Normal	“Hard contact”			
5.	Tangential	Penalty with friction coefficient of 0.3	5.16 (The reference)	4.06 (The reference)	-0.11
	Normal	“Hard contact”			
6.	Tangential	Static-kinetic exponential decay	5.18 (0.39 [%])	4.06 (0.00 [%])	-0.10
	Normal	“Hard contact”			

Table 15: Results of the different contact rules.

The Figure 45 highlights the deflection of the beam during this exercise as well as the distribution of the stresses in the beam. It is important to note that a Deflection Scale Factor of 100 is used in order to visualize well the vertical displacement of the beam and its rotation in the contact area. Moreover, due to the use of this deformation scale factor of 100, the illusion

²¹ The stresses here considered in the beam are the ones in the central area, not the stresses observed at the supports of the beam.

²² The displacement here considered is the one in the z-axis (or also called U3) as visible in the Figure 47.

is given in the contact area that the beam penetrates the column. For test 1., the Normal-Hard contact contract rule is selected. Here, the beam and column do not enter each other but instead, their respective finite elements deform in the contact area.

The axes considered in Abaqus for this exercise are visible in the Figure 45. The maximum vertical displacement of the beam (approximately around the mid-span) varies from 4.91 [mm] to 5.32 [mm], a significant difference of 7.71 [%].

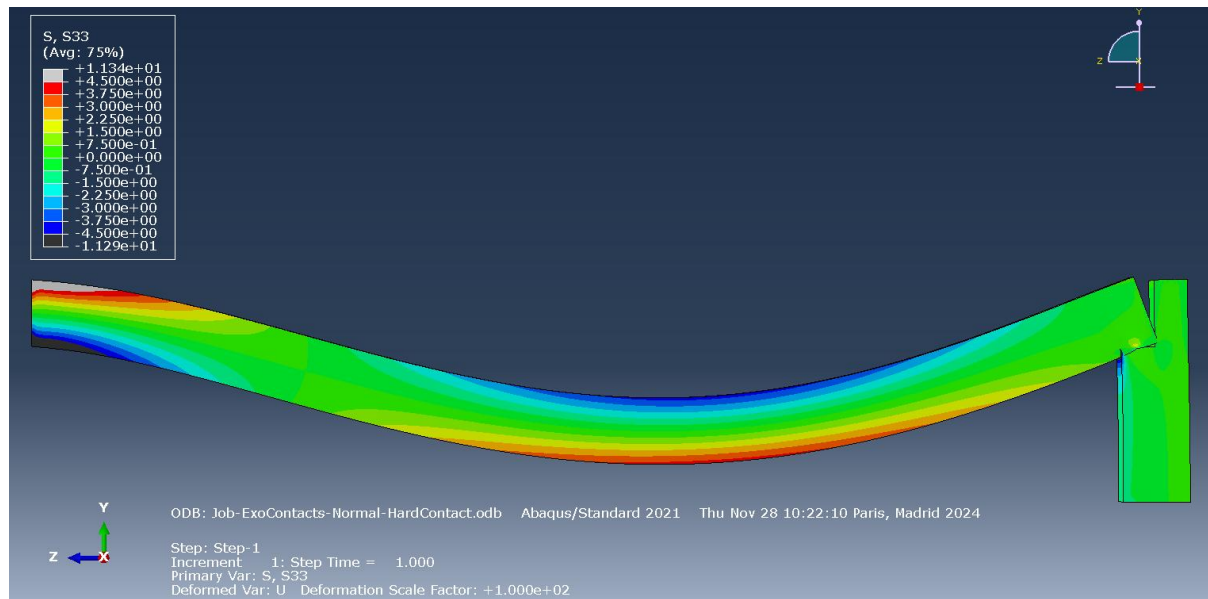


Figure 45: S33 stresses of the beam and column (test 1). Deformation scale factor: 100.

The biggest deflection of the beam is observed the **test 1.** and **3.** due to their Tangential-frictionless contact rule. In fact, beam element slides freely on one of its extremities, just like a roller support. The **test 3.**, with a Normal-Linear contact rule, shows a very slightly higher deflection due to a high stiffness coefficient implemented, allowing for little penetration in the material. A frictionless contact rule is not really pertinent for wood due to the friction induced by the fibres. Only specific cases, for example with a surface treatment like a thick paint, could justify a frictionless contact rule.

The “Rough” contact rule (**test 2.**) on the other hand, blocks the element from sliding and is therefore closer to a hinge. This contact rule could be pertinent for a wood assembly in the case where sliding is impeached but separation is possible.

The **test 4.** with a Cohesive contact rule showcases the lowest deflection at 4.91 [mm]. This is due to the contact rule which, here, impeaches the two surfaces in contact to separate from each other as visible in the Figure 46. This makes the right support closer to a clamped support on the area of contact, therefore reducing the deflection of the beam. The maximum nominal stress is chosen as 4 [MPa] for normal stresses only and 3 [MPa] for shear stresses.

This contact rule would be pertinent if elements of wood were glued together.

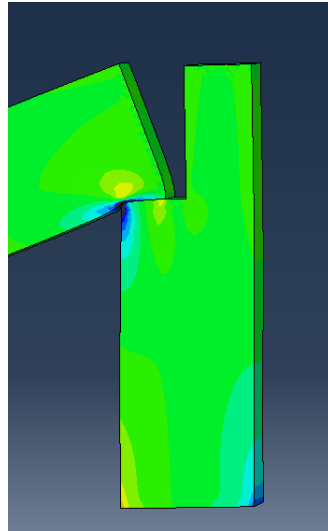


Figure 46: Test 4 with a cohesive contact rule. Deformation scale factor: 100.

The **test 5**. considers a friction coefficient tangentially simulating fibres intertwining and rougher finishes of the wood elements. It showcases a deflection of 5.16 [mm], more or less in the middle of the tests. This test accurately simulates the exercise chosen for Section 2.5. If an experimental test had to be reproduced, the friction coefficient could be calibrated.

Lastly, the **test 6**. uses a Static-kinetic exponential decay contact rule which models a passage from a static friction coefficient to a kinetic friction coefficient once the movement is underway. A static coefficient of 0.4 is chosen and a kinetic coefficient of 0.3, together with a decay coefficient of 10 [mm⁻¹]. The resulting deflection is close to the previous test with 5.18 [mm]. This contact rule is also modelling accurately what can happen between elements of wood. However, it is more complex than the **test 5**. and would require more calibration.

The maximum stress in the beam varies from 3.96 [MPa] to 4.09 [MPa], a difference of 3.18 [%]. Here, when comparing the stresses in the beam between the various tests, it was decided to consider only the stresses around the centre of the beam and not at its supports. With no surprise, the tests showcasing a higher maximum stress are the ones showcasing a higher mid-span deflection of the beam.

As visible in the last column of the Table 15 and in the Figure 47, the displacement of the column is highly influenced by the contact rules. The tests 1 and 3 showcase a displacement of the column towards the left (in the direction of the beam). The tests 2, 4, 5 and 6 showcase a displacement of the column towards the right.

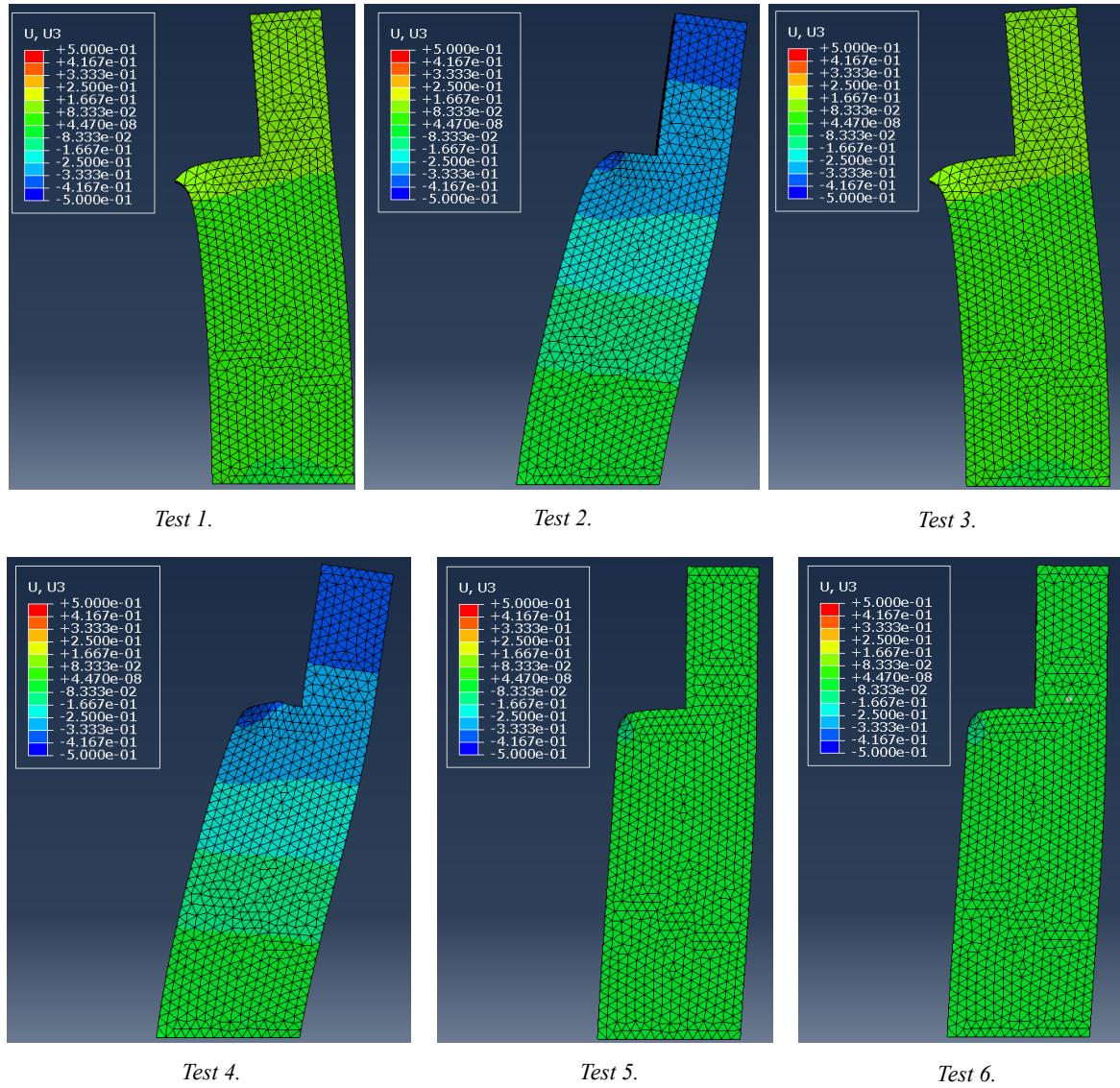


Figure 47: Displacement along the axis of the beam (z axis) for the 6 tests performed. Deformation scale factor: 500.

In order to explain these phenomena, four elements are to be considered. First, the parameter changing between the tests is the contact rule applied. Second, the beam undergoes a rotation of its right extremity due to the bending on the beam. As visible in the Table 16, the top fibre of the beam, at its right extremity, is displacing towards the left (towards the centre of the beam). The medium fibre, as globally admitted analytically, is also displacing towards the left but less strongly. The bottom fibre of the beam, on the other hand, is displacing towards the right. Finally, the resultant of force of the beam on its right support (the column) should be taken into account. The resultant's location is at the left extremity of the column.

The difference of displacement of the column between the tests can now be explained. The tests 1 and 3 are the only ones without a friction between the surfaces. The column, in these two tests, displaces towards the left because the resultant of the force of the beam is offset with the centre axis of the column.

	Displacement of the beam at its right extremity (next to the contact) [mm]		
Tests	Top of the beam	Neutral fibre	Bottom of the beam
1.	0.55	0.03	-0.56
2.	0.57	0.10	-0.41
3.	0.55	0.03	-0.56
4.	0.56	0.10	-0.33
5.	0.56	0.02	-0.50
6.	0.56	0.06	-0.50

Table 16: Summary of the displacements of the beam in the direction parallel to its longitudinal axis at its right extremity, next to the contact, for each contact test performed.

The tests 2, 4, 5 and 6 have their column displacing towards the right. They have the particularity of all having a friction either infinite or of a certain value. This displacement of their column can be explained by the relative displacement of the beam in its right extremity and the partial or complete liaison that happens between the beam and the column. The tests 2 and 4 have their column strongly displaced towards the right as the friction coefficient with the beam is infinite (rough and cohesive contact rules). The tests 5 and 6 have a lower displacement of their column as the friction coefficient with the beam is not infinite.

It is important to note that the chosen contact rules need to be adequate with the experimental situation or the joint of the building to be built. There is not one perfect contact rule working for all the wood-wood joints, the user must carefully choose it depending on the real contacts between the specimens. For the rest of this master's thesis, it was decided to use the contact rules from the **test 5**. (Normal-Hard contact together with Tangential-Penalty with friction coefficient of 0.3). This represents relatively well the interaction happening between two elements of wood with relatively plane surfaces due to milling. The friction coefficient could be improved from experimental testing with the type of wood required and the finishes of the surface wanted. Therefore, the error due to this choice is here not quantified and caution should be taken.

The other contact rules are still important to understand as they can be used in other situations. For example, the cohesive contact rule could be pertinent in the event of a wood-wood assembly where glue is applied, or screws added, bringing a cohesion between elements of wood. Additionally, for improved accuracy of the modelling of the behaviour of wood, the damage and fracture contact rules should be studied in depth.

2.5.2. Tolerances

Key aspects of the subsection: This subsection builds on Section 2.5.1 Contacts to emphasize the necessity of defining a contact rule in Abaqus for handling interactions appearing during simulations. It examines the effects of small tolerances in a simple beam-column assembly, highlighting their implications and discussing the results. This also highlights that special attention should be given to the importance of this parameter in assembly design, particularly due to wood's hygroscopic nature.

In this subsection, the impact of tolerances between elements is studied. In fact, tolerances between elements are a key parameter for timber buildings as wood is a highly reactive material to changes of hygroscopicity as discussed in the section 1.5 about the properties of wood. As a reminder, when wood elements absorb moisture, their volume is going to increase. With low tolerances, some contacts may appear and new stresses be induced. The opposite phenomenon is also common: the drying of wooden element reducing their volume, increasing tolerances and possibly affecting the stresses, deflections, etc.

The consequences on the deflections and stresses depending on the tolerances used can also be important during accidental situations like earthquakes where the beams can move significantly and enter in contact with the columns at their tip, also enduring new stresses in the beams and columns of the building.

The exercise chosen to highlight this phenomenon is the same as considered for the subsection 2.5.1 about Contacts. The main difference here is that the tolerance, or spacing, between the tip of the beam and the top of the column is now reduced from 50 [mm] for the subsection “Contacts” to 0.1 [mm] for the current subsection “Tolerances” as visible in the Figure 48. The tolerance is here chosen to have a contact between the tip of the beam and the column after approximately 20 [%] of load is applied. The contact rule here chosen corresponds to the test 5. with a Tangential- Penalty with friction coefficient of 0.3 together with a Normal-Hard contact rule. As the contact rules are selected through “General contact”, any contact present from the beginning, or developing throughout the simulation, is considered.

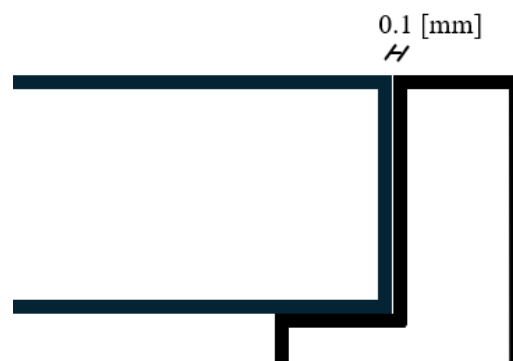


Figure 48: Tolerance of 0.1 [mm] between the beam and the column for the subsection “Tolerances”.

Additionally, a higher density of finite elements next to the contact area was chosen to extract better results. The top first part of the column (on a length of 400 [mm]) is given an approximate global seeding size of 10 [mm] (compared to 20 [mm] before). The right left of the beam, next to the column, is given the same approximate global seeding size for the last 250 [mm] as visible in the Figure 49. Additionally, the mesh in the rest of the elements is now coarser with an approximate global seeding size of 80 [mm] (compared to 20 [mm] before). This allows to keep a similar computational time as before, stay under the maximum limit of nodes of the Abaqus educational version and to have more accurate results in the contact area.

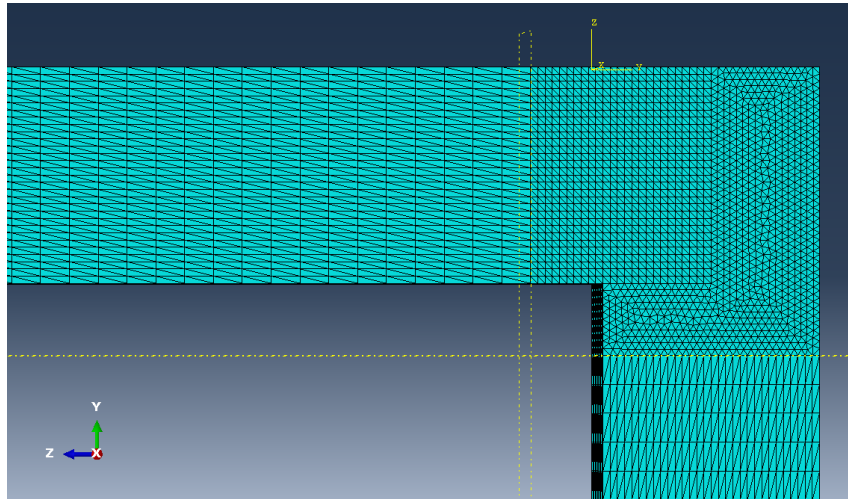


Figure 49: Lateral view of the higher-density mesh in the contact area.

The deflection at mid-span of the beam is now 4.83 [mm] compared to 5.16 [mm] previously (where the same contact conditions were used but a higher tolerance between the column and beam). In this new configuration, the right support of the beam starts as a roller support and becomes a hinge as soon as the tip of the beam enters in contact with the column (after approximately 20 [%] of final load is applied). The fact that the tip of the beam enters in contact with the column and induces a change in support conditions reduces the deflection at mid-span of the beam. This situation also induces stresses in the column as visible in the Figure 50 and a higher deflection of the column. The maximum stress in the beam is also reduced from 4.06 [MPa] to 3.95 [MPa].

The Figure 50 highlights the progressive distribution of the stresses in the beam and column during the loading. As the deformation scale factor is set to 1, the deformations visible on the images are the actual deformations. A gap progressively appears on the top-right corner of the beam as the beam undergoes an angular rotation at its right extremity. The black and blue colours represent the highest compression stresses in the elements whereas the grey and red represent the highest tension stresses. A concentration of stresses is clearly visible in the pivot area of the beam and column. Compared to the tests made in the subsection 2.5.1 about the contacts, here, an induced stress is visible in the column due to the tip of the beam entering in contact due to a low tolerance.

Using numerical analysis, it could be interesting to study the ideal tolerances of a certain wood-wood assembly depending on parameters like the moisture content strongly affecting the dimensions of the elements of wood as described in Section 1.5. This could help optimize the contacts between the elements while minimizing undesired additional stresses.

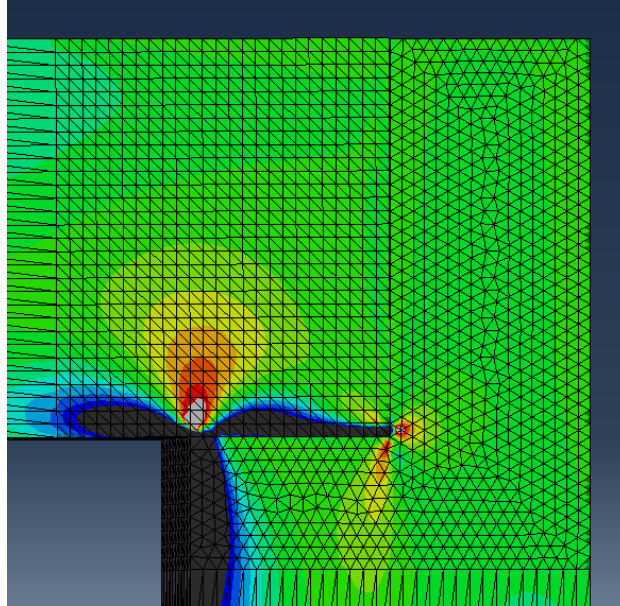


Figure 50: Stresses (S_{33}) in the contact area. Deformation scale factor of 1.

In conclusion, once the appropriate contact rules are selected, an absent contact at the beginning but appearing during a simulation can be modelled. This section allowed to confirm that as well as identify the consequences on the column and beam. The column is now subjected to an additional stress in its upper area whereas the beam is undergoing a lower deflection due to a change in its right support conditions: from a roller to a hinge. Additionally, the section highlights the importance of the tolerances in wood-wood assemblies.

2.5.3. Non-linearities

Key aspects of the subsection: Wood exhibits different plastic behaviours depending on the loading direction (perpendicular or parallel to the fibres) and type (tension or compression), resulting in either ductile or brittle responses. This section introduces a simplified approach to simulate wood's plasticity, essential for understanding phenomena observed in experiments or extreme events like earthquakes and high winds. Two tests are conducted: one with the original load from the previous exercise, showing negligible plasticity, and another with a load increased fourfold, clearly demonstrating plastic behaviour. While the model captures basic non-linearities, its limitation lies in the absence of fracture modelling.

In this subsection, the impact of non-linearities is studied. In fact, when wood is subjected to a load which overcomes the elastic domain of the material then plasticity appears. Due to the anisotropy of wood, wood has a different elastic and plastic domain for the following four main situations which are visible in the Figure 51: compression perpendicular to the grain, compression parallel to the grain, tension perpendicular to the grain and tension parallel to the grain. The characteristic strengths in the Figure 51 are for elements of wood free of defaults such as nodes or cracks. The characteristic strengths are denoted by a f-letter with the following meaning for the indices: 't' for tension, 'c' for compression, '0' for parallel to the grain, '90' for perpendicular to the grain and 'k' for characteristic. As a reminder, a characteristic strength is calculated in a laboratory on a test samples. The characteristic strength corresponds to the strength of the material, in the test situation, for which only 5 [%] of the tested samples have an inferior strength.

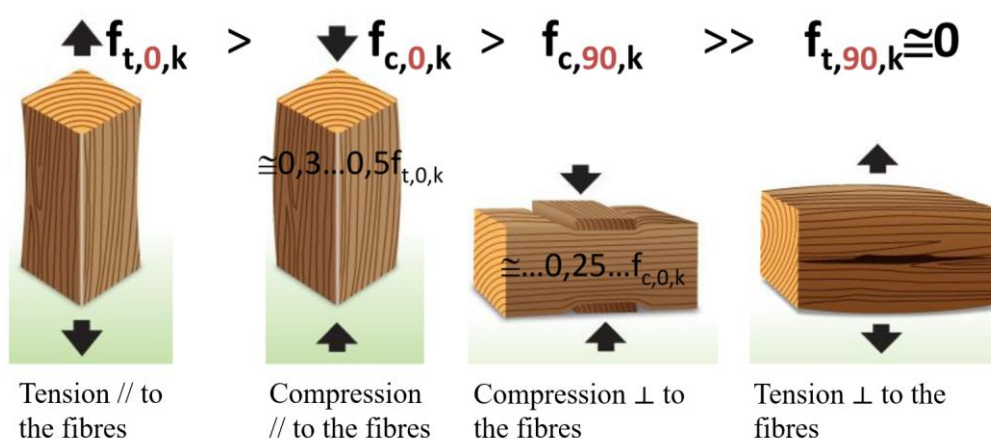


Figure 51: Characteristic strength in the four main loading scenarios of wood. (17)

Without taking into account defaults in the wood like nodes or cracks, the following behaviours are observed for each loading scenario, from strongest to weakest.

- **Tension parallel to the grain** usually behaves as elastic-brittle by breaking suddenly.
- **Compression parallel to the fibres** usually showcases an elasto-plastic behaviour where the fibres progressively separate from each other and buckle.
- **Compression perpendicular to the fibres** also usually highlights an elasto-plastic behaviour but the fibres are here progressively crushed.
- **Tension perpendicular to fibres**, with a low maximal strength, showcases a fragile behaviour with a separation of the fibres.

The order of strength of the first two (tension and compression parallel to the grain) is usually swapped due to defaults such as nodes which act like voids for tension loads parallel to the fibres and a strong local point for compression parallel to the fibres.

In this subsection, the test situation previously considered is taken. The material model, the contact rules and tolerances are kept the same. As a reminder, an orthotropic material model was used, the contact rule corresponds to a tangential-penalty friction of 0.3 together with a normal-hard contact rule. The tolerance between the beam and the column is of 0.1 [mm] to highlight the impact of contacts between them. As observed previously, contact is made on the horizontal part of the column (where the beam is simply deposited) as well as on the vertical part of the column where the beam enters in contact because of its rotation as well as sliding. The exercise is performed two times: one time with exactly the same pressure load as before and one time with four times the original load to have more finite elements entering plasticity.

To model the non-linearity of wood, a plastic Material Behaviour rule is created with the values visible in the Table 17. The reference scenario here considered is compression parallel to the grain. In Abaqus, it is necessary to choose a reference strength against which the other strengths are expressed as a percentage of the reference strength.

	Yield stress [MPa]	Plastic Strain
1	21	0
2	22	0.008

Table 17: Plastic rule considered in Abaqus.

The approximate yield here considered (passage from elastic to plastic) for the compression parallel to the fibres is taken at 21 [MPa] as a Douglas C24 is considered. A plastic strain of 0.008 is taken at 22 [MPa] therefore accounting for a slight hardening. Afterwards, a

“Potential” suboption is used to take into account the plastic behaviour of the material in the other directions. This suboption allows to take into account six Yield stress ratios

The obtention of the yield stress ratios is hereby described. First, the three main axes of the wood must be chosen. Here, as discussed previously, a circular orthotropy is considered. A piece of wood is taken (see Figure 52) to highlight the three main axes and the strength in all the directions for each face of the piece of wood. As before, the following axes are considered: the radial direction (1), the tangential direction (2), the longitudinal direction (3). The σ represent the yielding strength (or “elastic strength”) in a specific direction on a certain face of the wood. The subscript of these yielding strengths is made as follow: the first digit denotes the normal axis to the face considered and the second digit denotes the axis parallel to the direction of the strength considered.

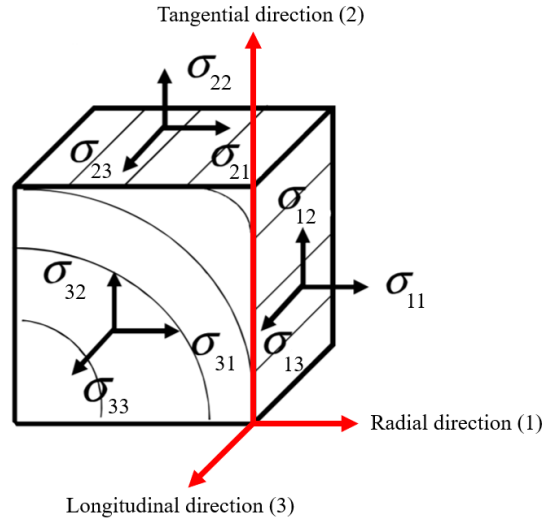


Figure 52: Main axes considered and the yielding strengths of the faces in the three mutually orthogonal directions. Adapted from (42).

Afterwards, six yielding strengths are here needed to be measured in order to calculate the yield stress ratios calculated with the Equation 6.

$$R_{11} = \frac{\sigma_{11}}{\sigma_0}; \quad R_{22} = \frac{\sigma_{22}}{\sigma_0}; \quad R_{33} = \frac{\sigma_{33}}{\sigma_0}$$

$$R_{13} = \frac{\sqrt{3}\sigma_{13}}{\sigma_0}; \quad R_{11} = \frac{\sqrt{3}\sigma_{23}}{\sigma_0}; \quad R_{11} = \frac{\sqrt{3}\sigma_{12}}{\sigma_0}$$

Equation 6: equations to calculate the six yield stress ratios necessary in Abaqus.

Where each yielding strength denotes the following meaning (42):

- $\sigma_{11}, \sigma_{22}, \sigma_{33}$: yielding tension/compression strengths in the radial, tangential and longitudinal directions, respectively.

- σ_{13} , σ_{23} , σ_{12} : yielding shear strengths in the radial section, tangential section and transversal section, respectively.
For example, the directions “13” denote the radial-longitudinal plane also called “radial section”.
- σ_0 : represent an arbitrary value used to obtain 1 for the reference yielding strength and below 1 for the other yielding strengths.

The following yielding strengths (Table 18) come from the literature (43) and were measured on a species of wood with similar elastic properties to the C24 Douglas here considered. They can be obtained from six distinct tests performed on clear specimens of wood as described in the Annex 5.2 Determination of the orthotropic parameters. The yield stress ratios calculated with the Equation 6 are described in the Table 19.

Yielding strengths		
σ_{11}	σ_{22}	σ_{33}
4.2	4.2	21
σ_{13}	σ_{23}	σ_{12}
12.12	4.24	2.42
σ_0		
21		

Table 18: Yielding strengths in six key directions.

Yield stress ratios		
R11	R22	R33
0.2	0.2	1
R12	R13	R23
1	0.35	0.2

Table 19: Yield stress ratios considered in Abaqus.

It is important to note that the plasticity model here considered in Abaqus is a simplification of the real behaviour of the wood. In fact, the main limit of this model is that the rupture of fibres is not modelled. Indeed, when fibres enter plasticity in real life, for example in tension perpendicular to the fibres, the separation of the fibres is abrupt, and a complete loss of strength happens in this area.

In the currently considered Abaqus model, when the fibres enter in plasticity, for example in tension perpendicular to the fibres, the fibres start to displace much more than they would if they were still in their elastic domain. Consequently, the deformations of the fibres in

the numerical model might be close to the deformations obtained experimentally. However, the strength of the fibres in the numerical model after entering plasticity is non-zero whereas it is very close to zero in real life situations for this scenario.

A perspective of improvement for future studies is to use in the contact rules the “Damage” and the “Fracture criterion” options. This was here not used as it requires a higher knowledge and experience in numerical modelling compared what was here possible to acquire. Additionally, it can also be advised to the interested readers and for future research to improve the numerical model by using Abaqus Subroutines and element deletions.

Another perspective of improvement of the model would be to perform a small experimental campaign on the desired wood planned to be numerically tested to improve all the material parameters (elastic and plastic) and therefore reducing the error in the modelization of the behaviour of the material.

Additionally, the use of a failure criterion to assess the transition between the elastic domain and the plastic domain of wood can be pertinent. The most common failure criteria identified in the literature are described in the Annex 5.1. They are however reserved to more advanced users as a user subroutine, or a plug-in, is necessary.

The results obtained for the original load (test a.) are first analysed. As visible in the Table 20, the deflection at the mid-span of the beam is 4.85 [mm]. This corresponds to a very slight increase of 0.02 [mm] compared to the exercise performed in the subsection 2.5.2 Tolerances.

Surface load applied		Deflection of the beam at mid-span [mm]	Maximum stress in the beam [MPa] ²³
a. F	0.036 [MPa] (The reference)	4.85 (The reference)	3.91 (The reference)
b. 4F	0.144 [MPa] (x4)	20.19 (x4.16)	16.10 (x4.12)

Table 20: Key results obtained in Abaqus.

²³ Like previously, the stresses here considered in the beam are the ones in the central area, not the stresses observed at the supports of the beam.

This is due to the fact that the load applied on the beam is relatively small and that only the finite elements visible in the Figure 53 have entered plasticity. To visualize which finite elements have entered in plasticity, the field output “AC YIELD” from Abaqus can be used. Values above 1 signify that the finite element has a stress state that exceeds the yield surface meaning plasticity. An AC YIELD of exactly 1 means that the stress state is exactly on the yield surface, the transition from elastic to plastic. A value below 1 indicate that the finite element is within elastic range. Therefore, here, the finite elements in plasticity are depicted in grey. The finite elements in red represent elements for which most integration points have entered plasticity but not all of them. We can approximate these finite elements as elements in plasticity. Here, the finite elements in plasticity correspond to the bottom area of the beam where the rotation happens with the column.

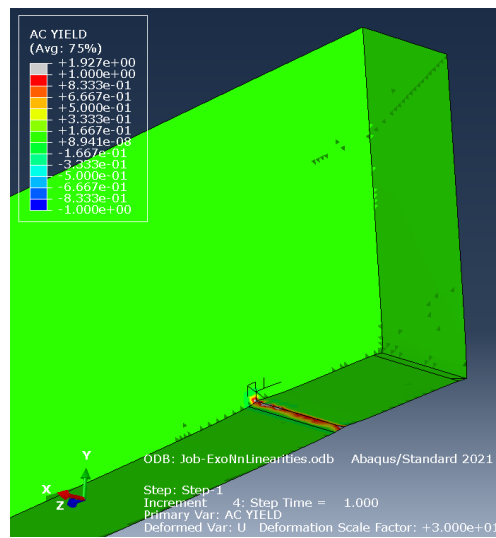


Figure 53: Highlight of the finite elements which entered plasticity (in grey) (test a., Deformation scale factor: 30).

The results obtained for the test b. (with four times the original load) are also visible in the Table 20 and showcase a deflection of the beam at mid-span of 20.19 [mm]. Even though the load is multiplied by 4, the increase in the deflection is multiplied by 4.16. This is due to the non-linearities appearing in the beam and in the column.

The Figure 54 shows the stresses (S33) in the contact area for the tests a. and b. The grey and red represent the highest tensile stresses whereas the black and blue represent the highest compressive stresses. It is interesting to observe the increase of stresses at the contact area in the column, especially in the area highlighted by an orange rectangle. The bottom tip of the beam enters in contact with the column where it pushes but most importantly, rotates upwards. This rotation induces the two grey bulbs of tension in the material. Slightly upwards of that, a small area in the column is enduring the compression due to the rotation of the beam.

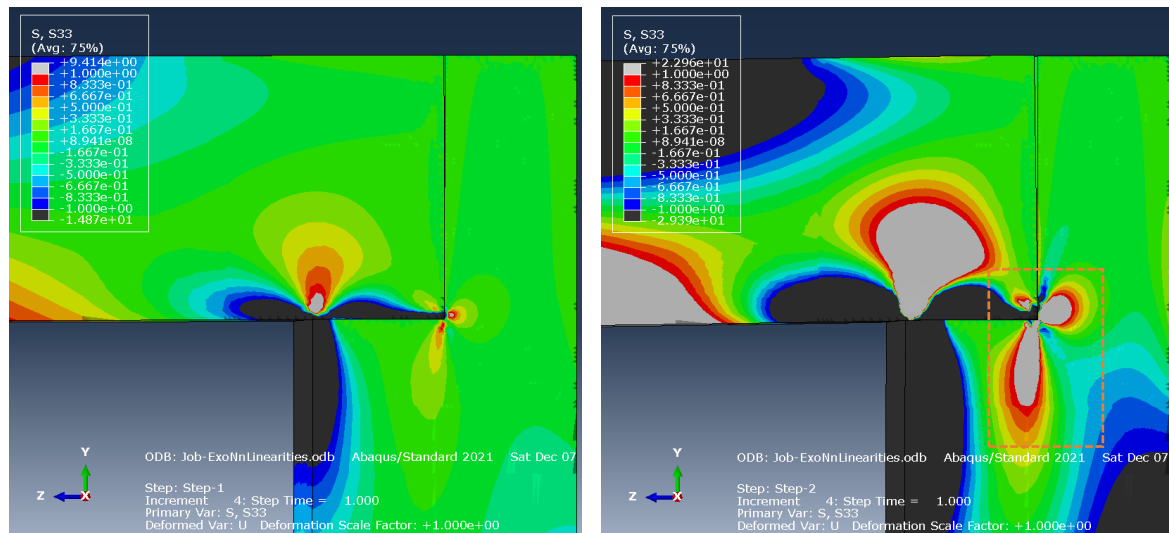


Figure 54: Stresses (S_{33}) in the contact area. Left: test a. for the original load. Right: test b. with the load multiplied by 4. (Deformation scale factor: 1)

The Figure 55 highlights the finite elements which have entered plasticity for the test b. (in grey and red). The left image represents the clamped side of the beam and cautious is advisory as accuracy of the results is limited next to the boundary condition (here a fully clamped surface). The central image represents the opposite side of the beam where the beam rests and rotates on the column. The main two contact areas are easily identifiable. The right image concerns the column where the two main contact areas are also easily identifiable. All these elements in plasticity endure a higher deformation therefore contributing to a higher mid-span deflection of the beam and better representing the behaviour of wood. The Figure 56, with a deformation scale factor of 50, highlights the local crushing of the fibres in the contact areas on the column.

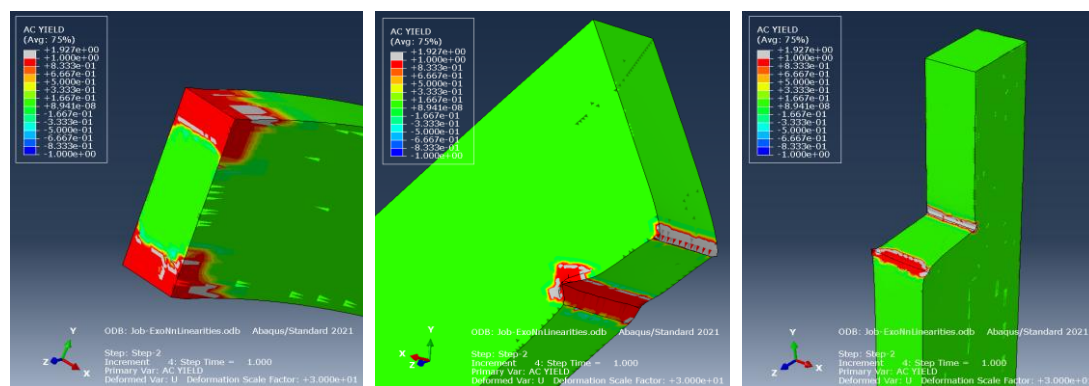


Figure 55: Elements which have entered plasticity (in grey) for test b. Left: clamped side of the beam. Centre: right side of the beam. Left: top of the column. (Deformation scale factor: 30)

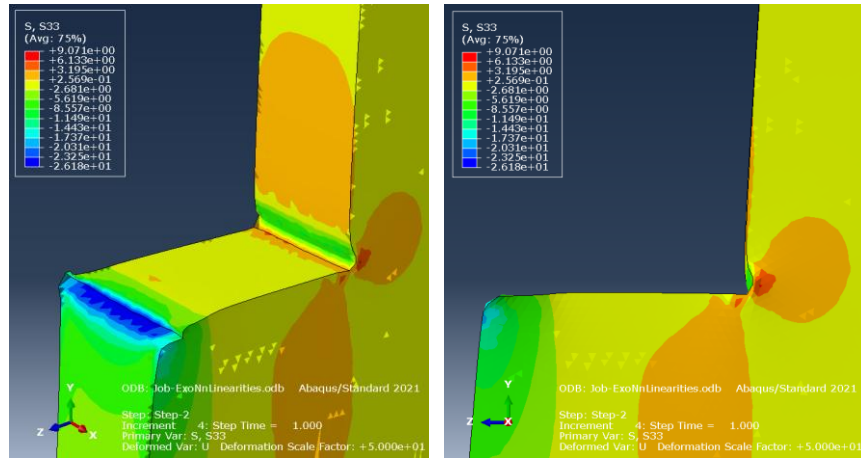


Figure 56: Stresses (S_{33}) in the column for the test b. (Deformation scale factor: 50)

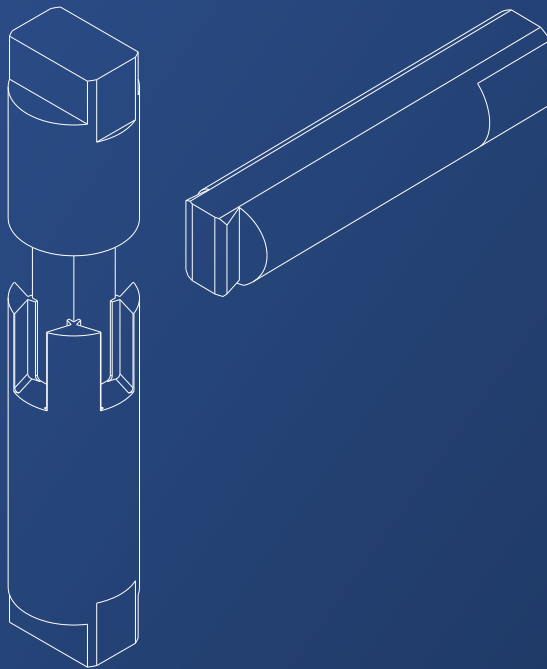
However, it is important to keep in mind that the current model does not account for “element deletion” of finite elements which would have reached a certain threshold like real wood fibres. For example, as mentioned previously, wood in tension perpendicular to the fibres showcases a brittle behaviour where the fibres separate from each other and the separated fibres therefore do not contribute anymore to the strength of the element. The apparition of cracks which is common in reality for timber subjected to high stresses is also not represented in this model which stays an approximation of the material. For interested readers, there exist complex possibilities of element deletions and the use of user subroutines to model more accurately the behaviour of wood past its elastic domain.

It is important to note that the current numerical model is still more qualitative than quantitative. An experimental campaign on the specific type of wood desired would allow to obtain more accurate results for the elastic domain as well as the plastic domain of the wood as described in this subsection. Moreover, the exercise here simulated could be reproduced experimentally and the use of a Digital Image Correlation (DIC) in the area visible in the Figure 54 could allow to verify if such a phenomenon as observed numerically also happens experimentally.

Page intentionally left blank for printing purposes.

Chapter 3

RoboTronc's assembly



3. RoboTronc's wood-wood assembly

Chapter three begins by presenting an experimental pull-out test conducted by the RoboTronc team on a wood-only column-beam dovetail assembly. The primary objective is to replicate this experiment numerically using the newly developed numerical model for wood. The chapter concludes with a comparison of the results from the experimental and numerical tests.

3.1. Experimental pull-out test conducted in July 2024

Key aspects of the section: This section details a pull-out experiment conducted by the RoboTronc team on a wood-wood column-beam dovetail assembly. The test aimed to determine the pulling strength of a critical column part and simulate movements caused by façade depressions due to wind or beam shrinkage from hygrometric changes. Analysis relies on the force-displacement graph and accompanying figures. A first fracture occurs after approximately 4 [mm] of beam slip and 6 [kN] of force at the pulling actuator. Afterwards, the assembly shows ductility as it still resists up till a maximum strength of 18 [kN] where significant cracking now appears in a different location of the column. Wood quality and milling precision fell below expectations. Section 3.2 will perform the same test, numerically, to confront the quality of the current numerical model built.

3.1.1. Overview of the assembly and testing procedure

As part of the RoboTronc project, experiments have been conducted (not by the author of this present master's thesis) in July 2024. They correspond to the testing of a column-beam assembly made of Douglas logs and machined using CNC milling machines with a 6-axis robotic arm. This kind of assembly could be seen in a building. Technical drawings (and 3D perspectives) describing the column and beam considered can be seen in the Figure 57 till the Figure 59. All the dimensions on the drawings are expressed in [mm]. The dimensions here described are the ones aimed for the experimental test and are not necessarily the dimensions that would be used in a building.

Other beam and column assemblies have been studied by the RoboTronc team but are not covered here as this master's thesis focuses exclusively on the numerical modelling of the selected assembly. This particular assembly was chosen for its compatibility with robotic milling tools available at the sawmill, its lack of steel components and other advantageous properties. Additionally, more experiments are to be conducted by the RoboTronc team and their results could be used by future researchers, including as a mean of comparison and improvement of the numerical model here developed.

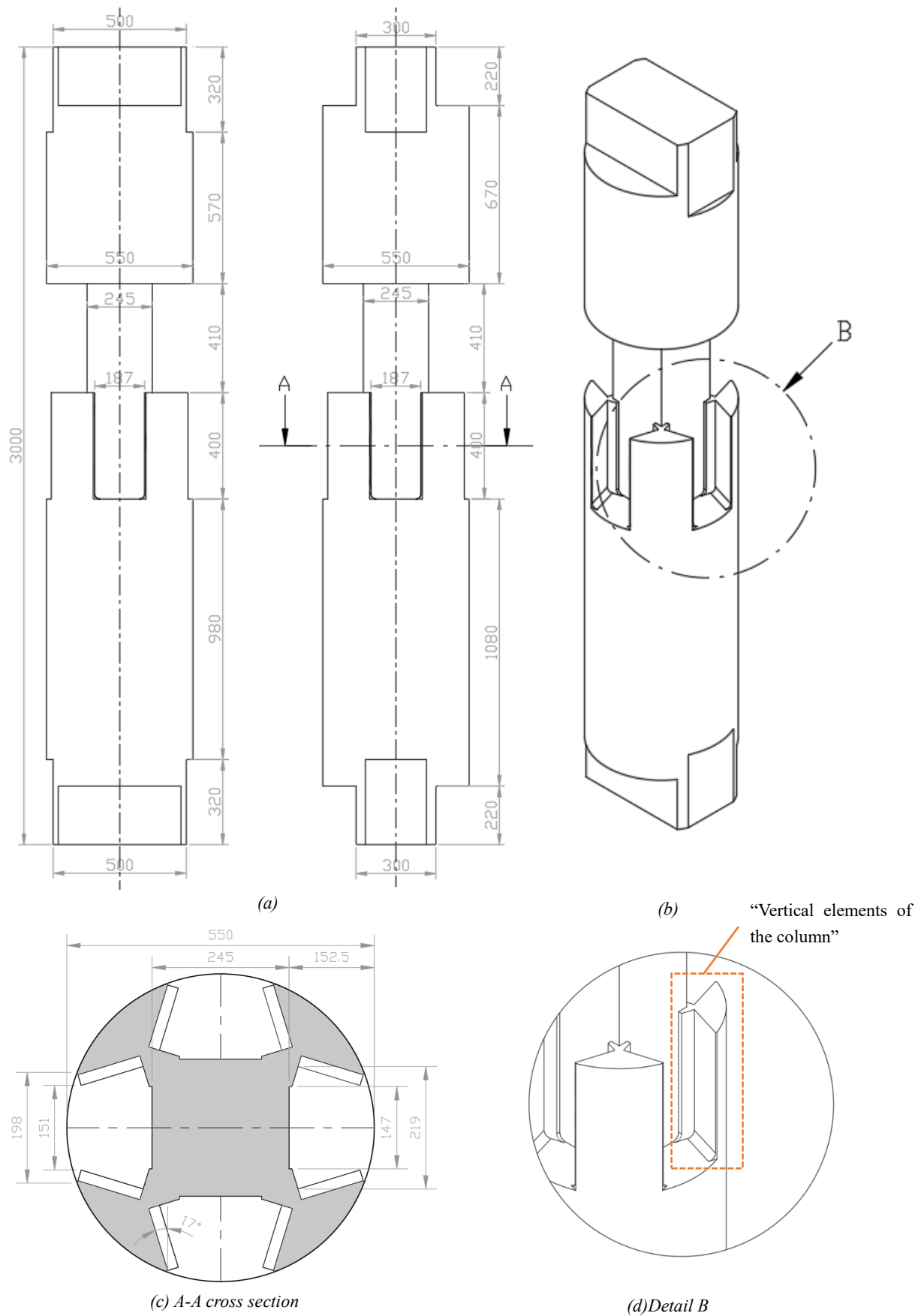
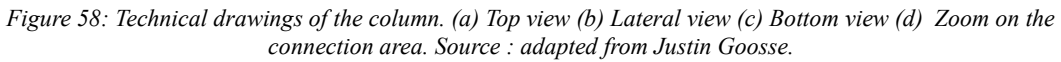


Figure 57: Technical drawings of the column. (a) Lateral view (b) 3D perspective (c) A-A cross-section (d) 3D perspective zoom of the connection area (detail B). Source : adapted from Justin Goosse.



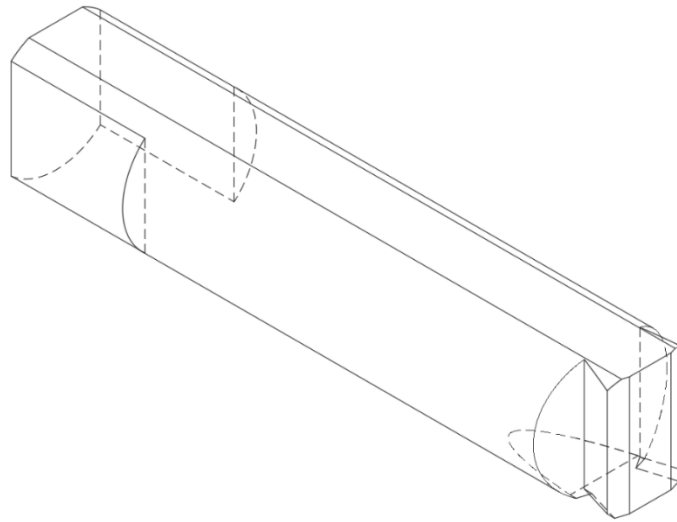


Figure 59: 3D perspective of the beam.

Globally, the circular timber column here considered is 3 [m]-tall with diameter of 550 [mm] with the connection taking place approximately at half its length. The column is made to accommodate four beams but only one is here considered as two axes of symmetry are visible and the effects can be deduced. The beam is 2 [m]-long and has a diameter of 450 [mm] at its widest. The Figure 60 highlights the complete geometry used for the experiments as well as the numerical simulations discussed in the subsection 3.2.

The column-beam assembly corresponds to a special dovetail assembly with adjustments made to a classic dovetail assembly in order to be robot compatible. The tolerance between the column and beam when assembled is visible in the Figure 61 and the Figure 62. It corresponds to a tolerance of 5 [mm] at the tip of the beam and 1 [mm] on the sides of the contact. The machined area in the column, above the beam, is necessary in order to slide the beam in the column.

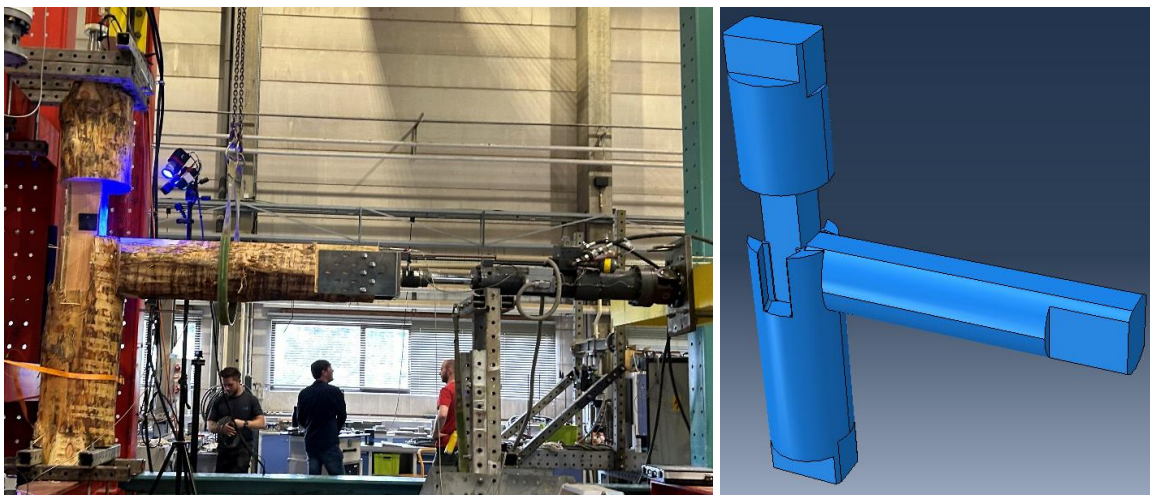


Figure 60: Geometry of the column and beam tested experimentally and numerically. Left: experiments in laboratory (source: Serena Van Nimwegen). Right: 3D model in Abaqus.

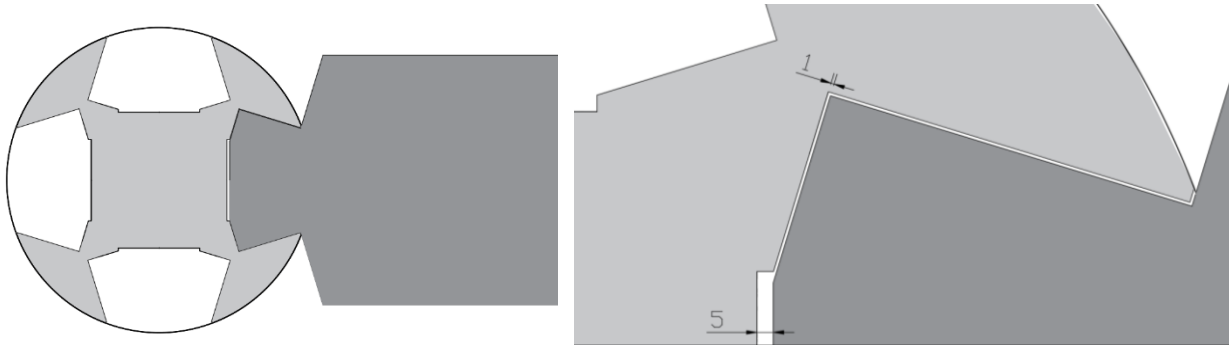


Figure 61: Top view of the column-beam assembly (column in light grey, beam in dark grey) and the tolerances used in the experiments and the numerical simulations.



Figure 62: Zoom on the dovetail assembly. Left: experiment in laboratory when the fracture has happened (source: Serena Van Nimwegen). Right: 3D model.

The top and bottom parts of the column are machined to allow the connection to the desired boundary conditions. In this case, the boundary conditions can be seen in the Figure 60 (left) and are schematized in the Figure 63. For the top of the column, the vertical face whose normal direction is parallel to the beam has its displacement blocked in its normal direction. For the bottom of the column, all the vertical faces are also blocked in their respective normal direction. The bottom face of the column is also blocked in its normal direction. Moreover, a load of 495 [kN] is applied by an actuator to the top of the column to simulate the load from above floors. Lastly, a U-shape steel plate is fixed with five screws on each side of the beam (see Figure 60) and linked to an actuator to perform a pull-out test. It corresponds to a progressive pull of 50 [mm] (using displacement control) of the beam along its longitudinal axis.

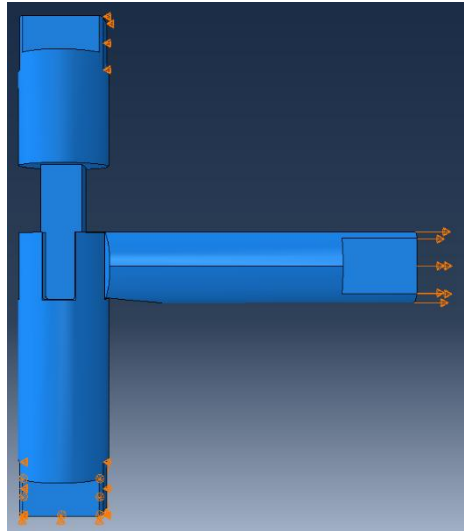


Figure 63: Lateral view of the 3D model with its boundary conditions.

The pull-out test is commonly performed experimentally for several reasons. First of all, this test allows to clearly identify the strength of the vertical elements of the column next to the contact area when they are being pulled by the beam. The “vertical elements” refer here in the text to the parts of the column which are holding the beams as highlighted in the Figure 57, (d). In a normal use case of a building, its beams and columns are loaded in flexure with an approximately uniform load. In this scenario, the beams are exercising a pulling force on the columns, so it is important to know the strength to expect from such assembly.

The test also allows to identify the failure modes and the behaviour of the assembly during this type of loading. Additionally, this test simulates what can happen to the assembly of the beams and columns of a building when wind induces a depression on a façade, when users are exercising a breaking force or when the beams are undergoing a reduction of their length due to changes in the hygroscopic conditions. The results here obtained could also be compared to a situation with a local reinforcement with screws, but this is not done here and can be the subject of future studies. It must be noted that the pull-out test is not simulating the entirety and complexity of the phenomena occurring in buildings.

To extract the results of the test, different tools were used such as Linear Variable Displacement Transformers (LVDT). They correspond to a type of electrical transformer allowing to measure linear displacements. They have been placed on the column next to the contact area (see Figure 64 (right)) as well as on the top of the beam between the tip and the actuator to see if slippage was occurring between the beam and the plate fixed to it. DIC (Digital Image Correlation) was also used next to the contact area as visible in the Figure 64 (left). This is why a white paint and black dots can be observed in the Figure 62. As mentioned earlier, it allows the measure of deformations or strains. Lastly, the displacement and the force at the pulling actuator was recorded. This will be our main tool to compare the results with the numerical simulations together with a visual comparison.



Figure 64: (left) View of the cameras required for the DIC. (right) View of the location of the LVDT's on the column (same on each side of the column).

Once everything is set up in place, including the load on the top of the column, the pulling from the actuator on the right can start as well as the measure of its force and displacement. Few interesting phenomena have appeared during the test and are described here. The state of the column and beam at the end of the experiment is visible in the Figure 65 and Figure 66.



Figure 65: Views of the cracks observed on the column after the test. Source: Serena Van Nimwegen.



Figure 66: Views of the cracks observed on the column. Source : Serena Van Nimwegen.

The load at the pulling actuator was applied in three main steps following the norm NBN – ISO 6891 (44) illustrated by the Figure 67 and described here.

- **Step 1:** a progressive load is applied up till 40 [%] of the maximal estimated load that the assembly can resist. Here, the maximal estimated load that the assembly can resist (denoted F_{est}) was estimated at approximately 8.5 [kN]. Therefore, at 3.4 [kN] (40 [%] of 8.5 [kN]), the increase of the load was stopped and maintained for 30 seconds. F denotes the force applied by the actuator at the instant t .
- **Step 2:** the load is now brought to 10 [%] (or 0.85 [kN] in this test) of the maximal estimated load that the assembly can resist and maintained for 30 seconds.
- **Step 3:** the load is now progressively increased up till an ultimate state or a desired slip is reached (here, approximately 50 [mm]).

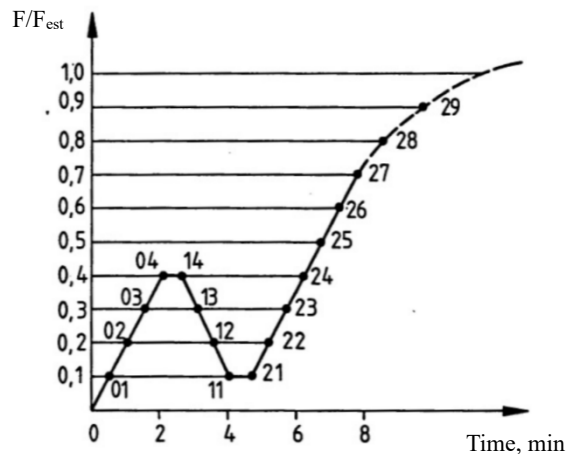


Figure 67: loading procedure described in the ISO6891. (44)

Theoretically, given the tolerances aimed for the experiment (of 1 and 5 [mm]) between the beam and the column, a distance of 3.27 [mm] in the axis of the beam should be necessary before the beam enters in contact with the vertical elements of the column, as visible in the Figure 68. During this time, only the friction between the horizontal part of the beam and column should happen. This should show as a 3.27 [mm] beam displacement with very little force applied (approximately 180 [N]). This is however not seen here due to the impossibility

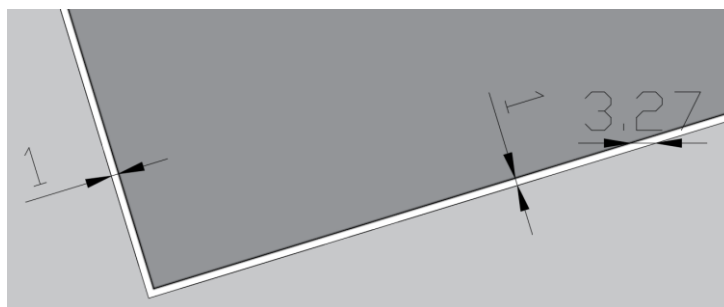


Figure 68: Zoom on the theoretical tolerances of the assembly column-beam.

of respecting such strict tolerances in real wood assemblies. The optimal tolerances for this assembly are still under study by the RoboTronc team.

3.1.2. Experimental results analysis and interpretation

The force-displacement graph obtained from the data of the actuator is visible in the Figure 69. Six main phases can be observed as highlighted by a dotted red rectangle and a number at its top. Each of them is described in detail down below.

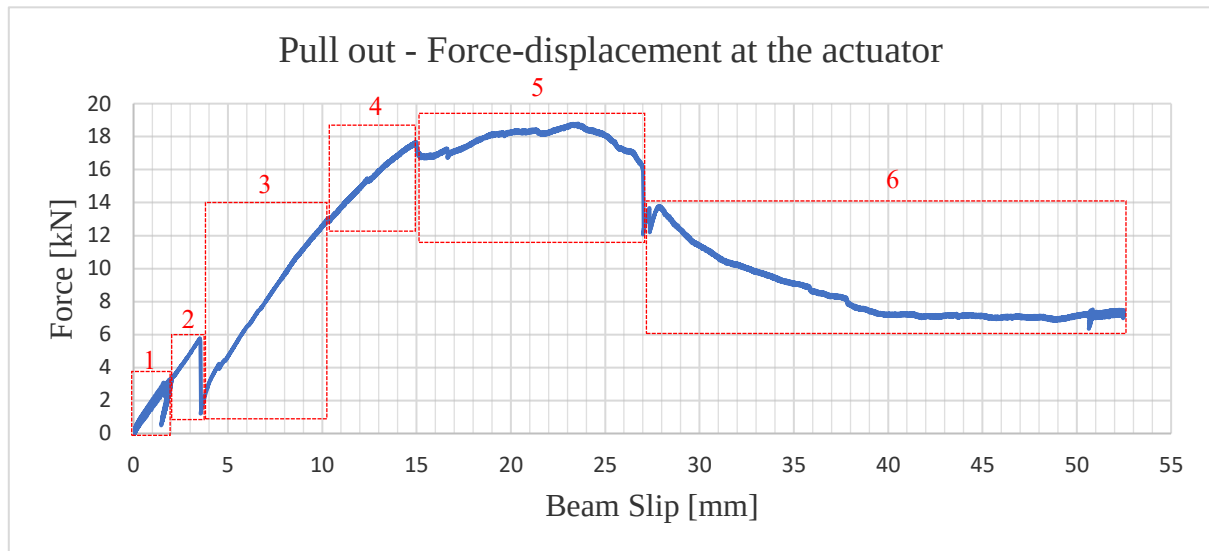


Figure 69: Force-displacement curve measured at the pulling actuator.

- **Phase 1** (0-2 [mm]): from the beginning of the test till 3.4 [kN] (2 [mm] beam slip), noise is observed in the force-displacement curve. It is here schematized by a thicker curve. This can be explained by irregularities in the configuration or quality of the actuator. At 3.4 [kN] and a slip of 2 [mm] of the beam, following the norm, the increase of the load is stopped and maintained for 30 seconds. Afterwards, the load is progressively reduced up till approximately 0.85 [kN] or (1.5 [kN]) and maintained at this level for 30 seconds. At this moment, the assembly remains mostly elastic with a small plasticity observed (due to small cracks) as the slope of the unloading is not perfectly parallel to the first loading phase.
- **Phase 2** (2-3.5 [mm]): now begins the progressive loading of the assembly until approximately 50 [mm] of beam slip. At approximately 3.5 [mm], the loading is stopped and drops from 5.7 [kN] to 1.21 [kN]. The displacement is not influenced. This phenomenon is due to the sudden cracking of the vertical elements in the area highlighted in the Figure 70.
- **Phase 3** (3.5-10 [mm]): from 3.5 [mm] (1.2 [kN]) of beam slip to approximately 10 [mm] or 13 [kN] at the actuator, the strength of the assembly is mostly due the area of the column (highlighted by a green rectangle in the Figure 71) lower than the assembly.) It functions similarly as a column, horizontally pulled at its top, and fixed on a part of its bottom length.

The slippage of the beam increases proportionally to the force in an almost perfectly straight line.

- **Phase 4** (10-15 [mm]): starting at 10 [mm] of beam slip, progressive cracks appear in the area highlighted in the Figure 71 and yield to a lower slope in the force-displacement graph informing of a reduced strength of the assembly.
- **Phase 5** (15-27 [mm]): at approximately 15 [mm] of beam slip (17.7 [kN]), the area of the column described in the Figure 72 showcases significant cracking as observed in the force-displacement graph at the beginning of the phase 5 and highlighted in the Figure 65. The cracks are deeper just below the beam's corner and observed only on the side of the vertical element who previously ruptured. From 15 [mm] till 27 [mm], an alternating behaviour is visible. First, a softening is visible during 3 [mm] after the new cracks appeared. Then hardening is visible till approximately 24 [mm] followed by a strong softening ended by other significant crack ruptures at approximately 27 [mm] strongly affecting the strength of the assembly.
- **Phase 6** (27-50 [mm]): a progressive decline in the force required to keep the beam to slip is observed. Ductility is visible in the assembly as the last force measured is approximately 7 [kN].

In conclusion, the experiment allowed to identify the failure area schematized in the Figure 70 due to the low strength of wood in tension perpendicular to the fibres and the small resisting area. It is interesting to note that the assembly ruptured only on one of the two sides and it corresponds to the side with the lowest cross-section in the rupture area. A second failure area was observed as schematized in the Figure 72. These two failure modes arise the possible relevance of locally reinforcing the column with screws, especially for the first cracking area.

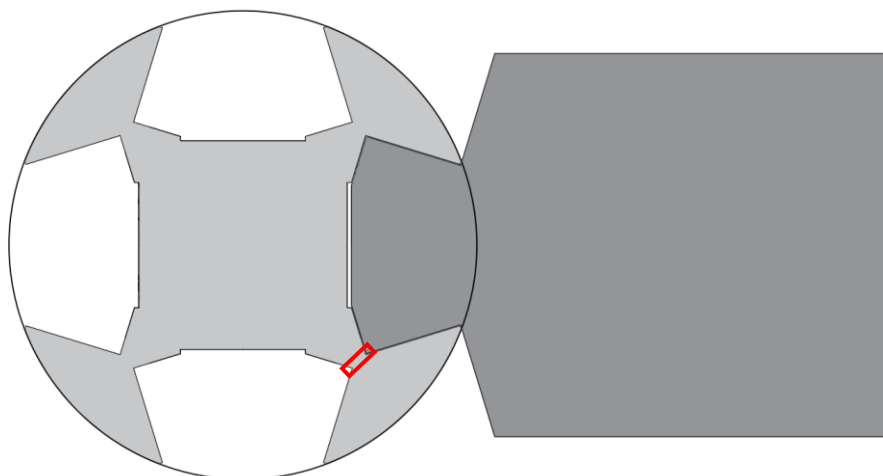


Figure 70: First area with cracks observed experimentally.

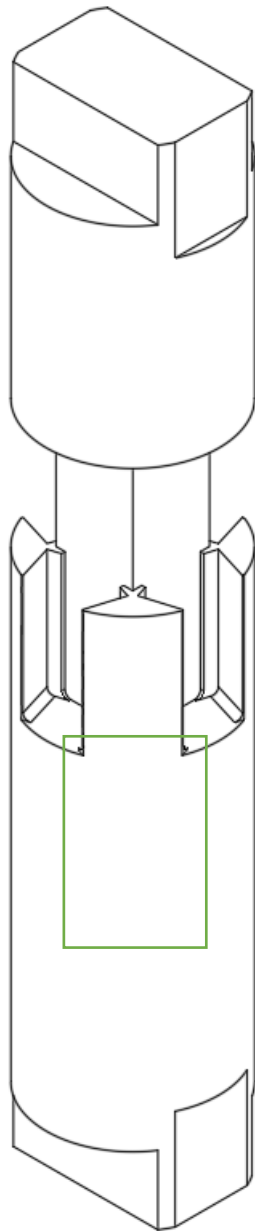


Figure 71: Area of the column resisting to the beam pulling once the left vertical element has ruptured.

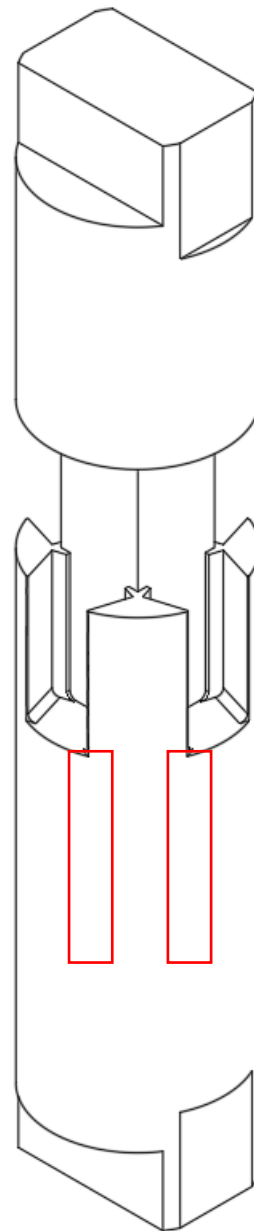


Figure 72: Area of the column cracking at approximately 15 [mm] beam slip.

3.2. Numerical simulation of the pull-out test

Key aspects of the section: The experimental test outlined in the previous section (3.1) is now replicated numerically using parameters developed during this master's thesis. The primary goal is to compare the numerical results with the experimental data to evaluate the model's strengths and limitations. Analysis relies on the force-displacement graph at the actuator and visualizations of plasticity, displacements, and stresses in the elements. Four phases are identified: beam slipping until contact, elastic assembly behaviour up to 25 [kN] and 10 [mm] slip, strength loss due to plastification, and a plateau highlighting a maximum strength of 35 [kN]. Section 3.3 will compare the results of the experimental test to the numerical test in detail to highlight differences and improvement opportunities.

In order to further assess the qualities and limitations of the numerical model developed till now in Abaqus, the experimental pull-out test described in the section 3.1 is here reproduced numerically. The steps to follow in Abaqus can be seen in Annex 5.4. Here, the key steps are discussed together with the results observed numerically and compared to the previously described experimental results in the section 3.3.

3.2.1. Numerical test parameters and configuration

First of all, the material model chosen here corresponds to the material model n°4 described in the subsection 2.3. As a reminder, it corresponds to an orthotropic material model for wood with circular coordinates. The Young moduli, Rigidity coefficients and Poisson coefficients taken are visible in the Table 21 and are an approximation of Douglas.

E_L [MPa]	E_R [MPa]	E_T [MPa]	μ_{LR}	μ_{LT}	μ_{RT}
11 000	770	570	0.292	0.449	0.39
G_{LR} [MPa]	G_{LT} [MPa]	G_{RT} [MPa]	μ_{TR}	μ_{RL}	μ_{TL}
720	880	80	0.374	0.036	0.029

Table 21: Young moduli, Rigidity coefficients and Poisson ratios considered for the numerical pull-out test.

For the modelization of the plastic behaviour of the wood, the parameters discussed in the subsection 2.5.3 about non-linearities are taken and reminded in Table 22 and Table 23.

	Yield stress [MPa]	Plastic Strain
1	21	0
2	22	0.008

Table 22: Plastic rule considered in Abaqus.

Yield stress ratios		
R11	R22	R33
0.2	0.2	1
R12	R13	R23
1	0.35	0.2

Table 23: Yield stress ratios considered in Abaqus.

The simulation is performed on half of the geometry as visible in Figure 73. This allows to significantly reduce the computational time (approximately in half) as the software is required to process less computation as the number of finite elements reduces, the contact areas are reduced and other boundary conditions. This is made only possible because an axis of symmetry runs along the centre line of the beam and column and Abaqus allows to specify such symmetry. It is interesting to note that the complete geometry can be displayed for the results to help understand the phenomena. The placement of the beam on the column is made with the same tolerances as aimed for the experiment and visible in the Figure 61, namely 1 [mm] tolerance between the sides of the beam and the column and 5 [mm] between the tip of the beam and the column.

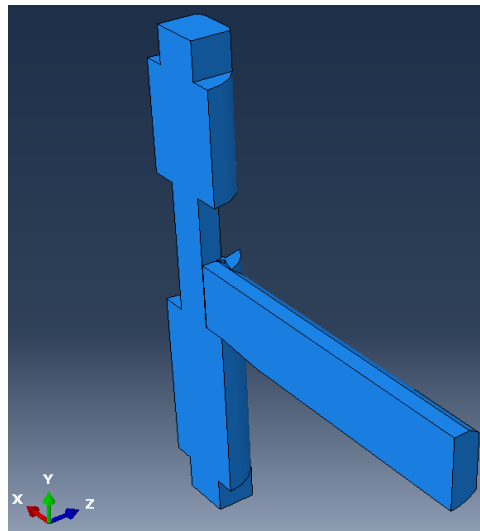


Figure 73: 3D perspective of the geometry tested numerically in Abaqus.

The contact rule here considered corresponds to the contact rule n°5 discussed in the subsection 2.5.1, Table 15. It corresponds to Normal-Hard contact rule together with a Tangential-Penalty with friction coefficient of 0.3.

The boundary conditions are the ones visible in the Figure 74. For the top of the column, the vertical face has its displacement blocked in its normal direction. For the bottom of the column, all the vertical faces are also blocked in their respective normal direction. The bottom face of the column is also blocked in its normal direction. Moreover, a load of 247.5 [kN] is

applied at the top of the half column to simulate the 495 [kN] applied experimentally on the top of the complete column. It simulates the load from above floors.

Additionally, the self-weight of the elements is taken into account. Lastly, the displacement induced by the actuator during the experiment described in the subsection 3.1 is modelled by a displacement control load of 20 [mm] in the longitudinal axis of the beam as visible in the Figure 74.

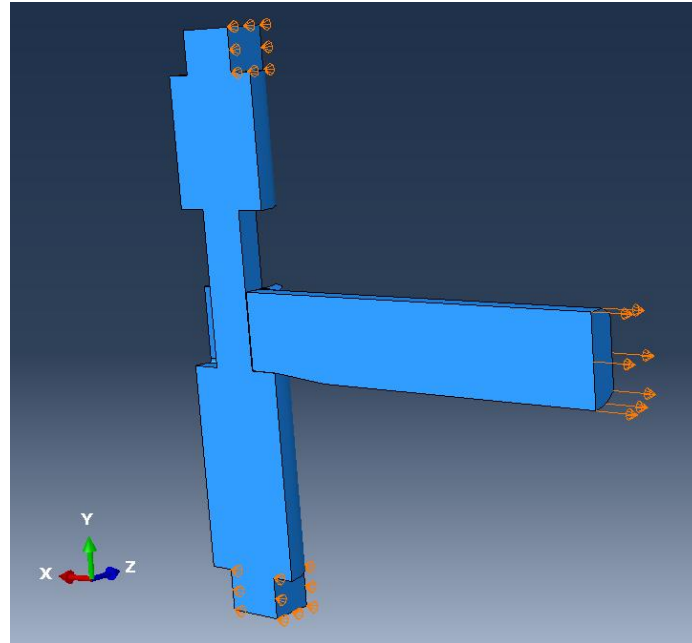


Figure 74: Boundary conditions considered numerically.

The finite elements used are linear tetrahedral (also called C3D4) and were chosen for their lower computational effort and good results with an appropriate mesh quality. For the beam, the “global size” of the seeding²⁴ is chosen as 100 [mm] whereas for lines next to the contact area, a “global size” of 20 [mm] is chosen as visible in the Figure 75. For the column, the same “global size” of seeding is chosen (100 [mm] for the area far from the contact and 20 [mm] for the area close to the contact as visible in the Figure 76). The final mesh considered for the beam and column is visible in the Figure 77 and Figure 78, respectively.

²⁴ As a reminder, the “global size” of the seeding corresponds to a non-strict constraint that Abaqus tries to respect. The size, in units of distance, corresponds to the desired distance between two “seeds” of the mesh. Seeds approximately correspond to nodes of the future mesh. Abaqus tries to respect it but can lower this constraint in some key areas where the geometry is more complex to increase the chances of a correct meshing of the geometry.

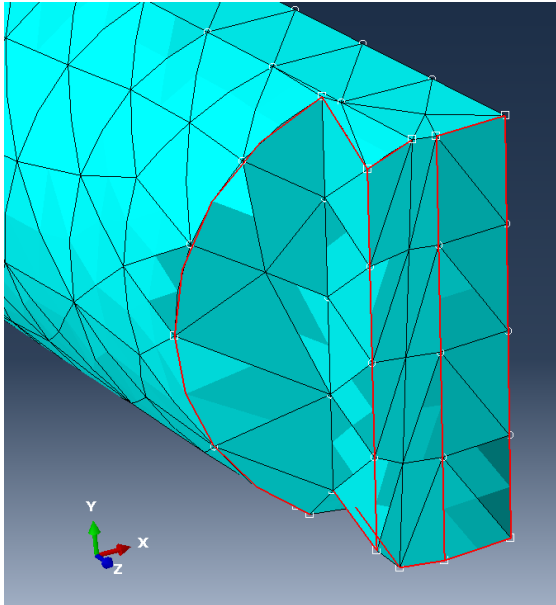


Figure 75: View of the mesh of the beam before local seeding. In red, lines where a local seeding is implemented for the beam.

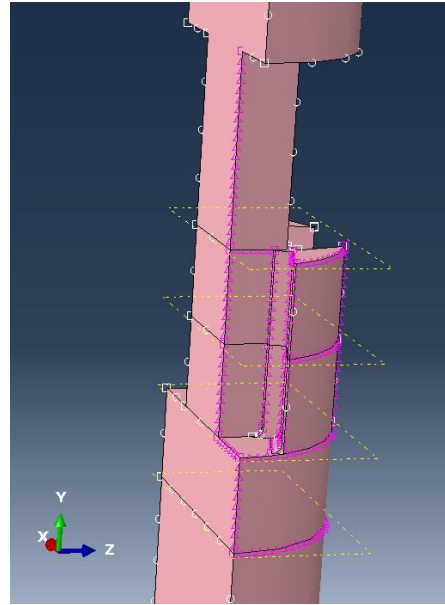


Figure 76: In pink, lines where a local seeding is implemented for the column.

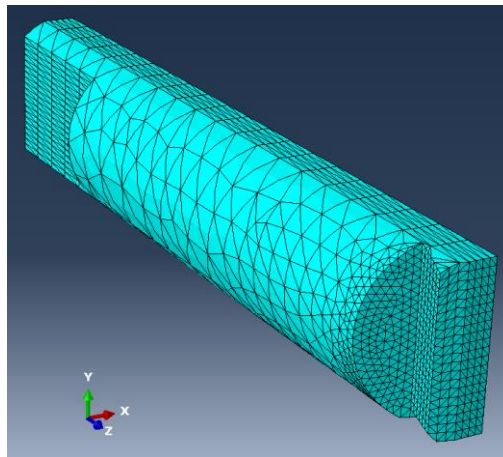


Figure 77: Final mesh on the beam.

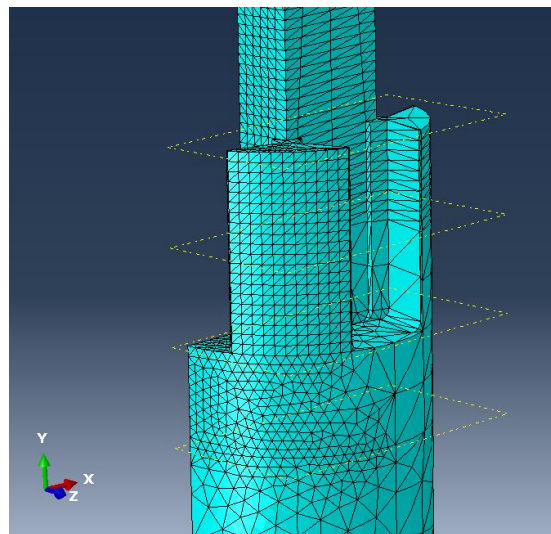
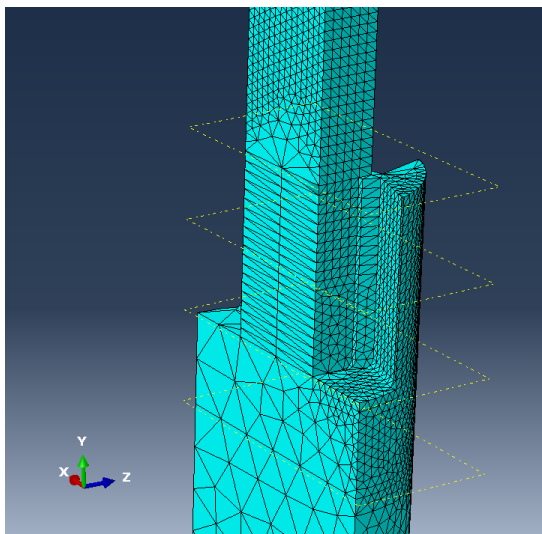


Figure 78: Final mesh on the column, in the contact area.

3.2.2. Numerical results analysis and interpretation

The Figure 79 highlights the movement performed by the beam with regard to the column. The force-displacement graph of the “numerical actuator” is plot in Figure 80 and showcases four main phases.

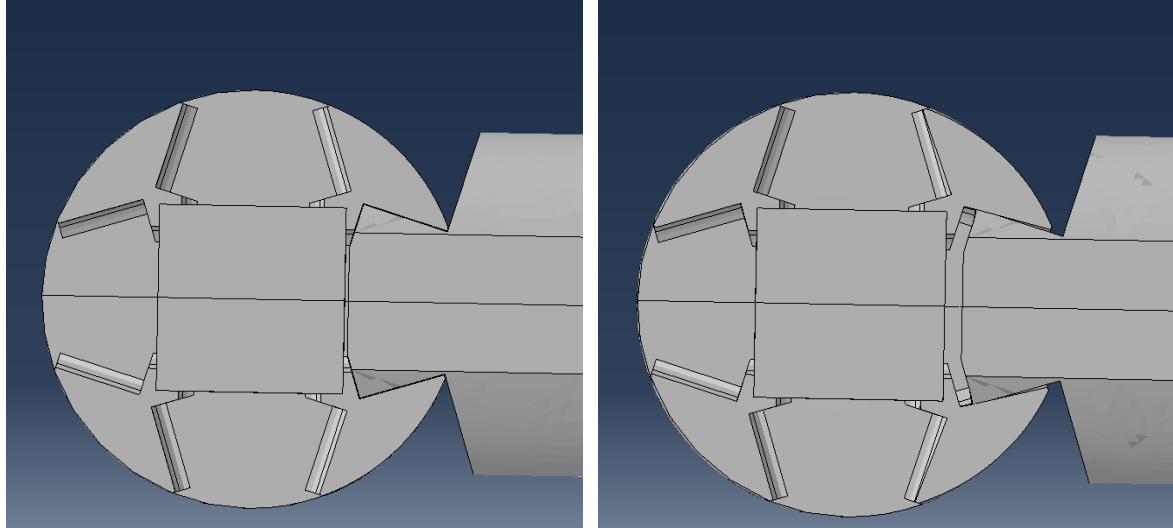


Figure 79: Top view of the dovetail assembly. Left: the initial position. Right: final position after 20 [mm] of beam slip.

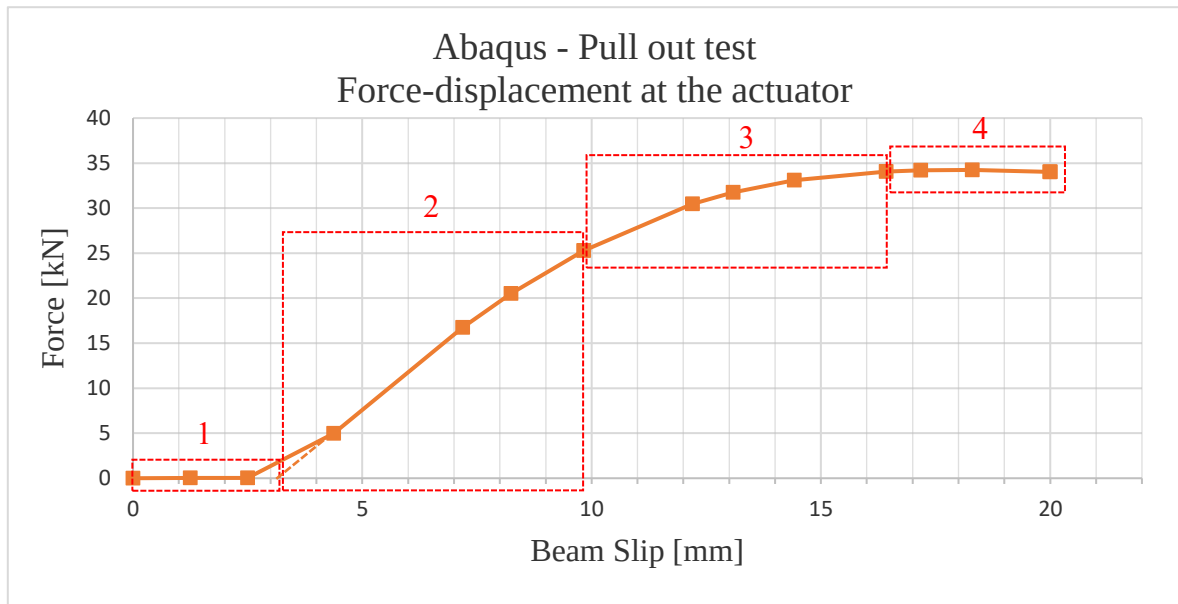


Figure 80: Force-displacement curve measured at the pulling actuator in Abaqus.

- Phase 1 (0-3.27 [mm]):** the first phase is observed from 0 [mm] to 3.27 [mm] and it corresponds to the slippage of the beam necessary for the contact between the beam and column to happen. Numerically, the spacing (or tolerance) between the column and the beam discussed previously (see Figure 68) is perfectly respected. The elements are perfectly plane and their positioning is accurate. A very low force is required to pull the beam at this moment as only the friction between the horizontal surfaces takes place.

- **Phase 2** (3.27-10 [mm]): From the instant the contact appears between the beam and the column till approximately 10 [mm] (or 25 [kN]), a mostly elastic behaviour is observed as the slope is mostly straight.
- **Phase 3** (10-16.5 [mm]): From 10 [mm] (or 25 [kN]) till approximately 16.5 [mm] (33 [kN]), the slope gets lower showcasing that a lower force is now required to obtain the same beam slippage. This is due to the progressive increase of finite elements reaching plasticity in the vertical elements of the column (in the area where the first cracks appeared experimentally)(see Figure 70). It is important to note that the numerical model here used does not simulate cracks and element deletions. Therefore, the cracks that were observed experimentally due to tension perpendicular to the fibres are here simulated by an increase of deformation and reduced strength compared to their elastic domain. This is a clear limitation of the model and a perspective of improvement.
- **Phase 4** (16.5-20 [mm]): Afterwards, from 16.5 [mm] till 20 [mm] (or 34 [kN]), the beam continues to slip with the simulated action of the actuator, but the force required for this pulling is now slowly decreasing till the end of the simulation. This showcases the strong loss of rigidity of the assembly. More precisely, the data showcases a peak of force at 34.24 [kN] at 18.3 [mm] beam slippage and a force of 34.02 at 20 [mm] beam slippage, therefore simulating the strength softening of the assembly under a high beam slippage.

The Figure 81 and Figure 82 highlight the finite elements which have entered plasticity (in red and grey) at the end of the test simulation for the column. The orange rectangles and numbers highlight the precise areas of fracture.

- The **area of fracture 1** is the most critical as it has a strong impact on the rigidity of the assembly against pull-out.
- The **area of plasticity 2** corresponds to a compression perpendicular to the fibres induced by the beam on the column.
- The **area of plasticity 3** appears as the vertical elements of the column endure a rotation around the y-axis together with a pull-out force pulling them away from the beam.
- The **area of plasticity 4** is due to the rotation that the vertical elements of the column endure.
- Finally, the **area of plasticity 5** is mostly due to a concentration of the stresses due to the vertical load (of 495 [kN]) and the geometry.

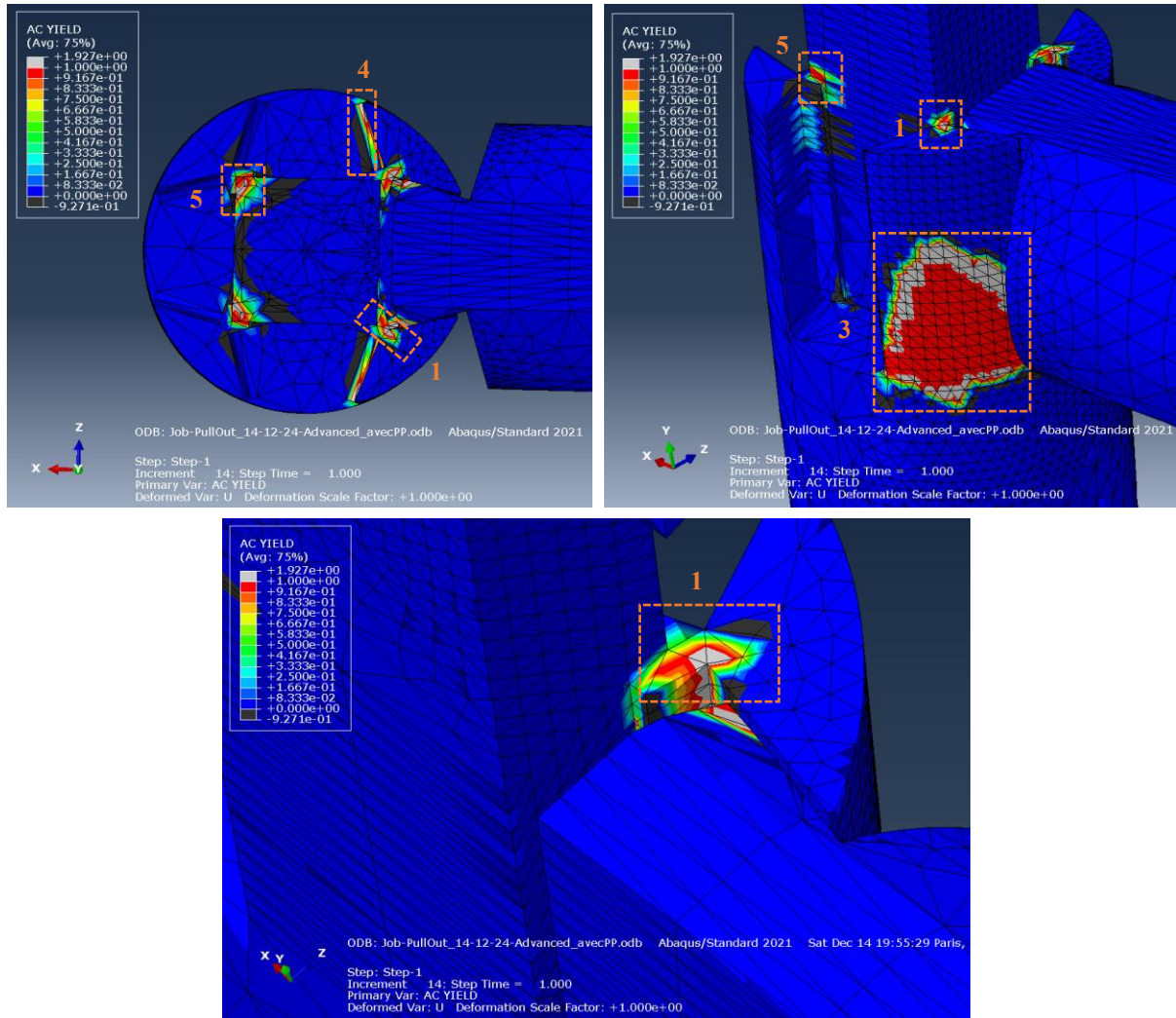


Figure 81: In red and grey, finite elements which have entered plasticity. Orientation of the views visible through the triad visible on the bottom left of the figures. The numbers and squares in orange highlight the fracture areas.

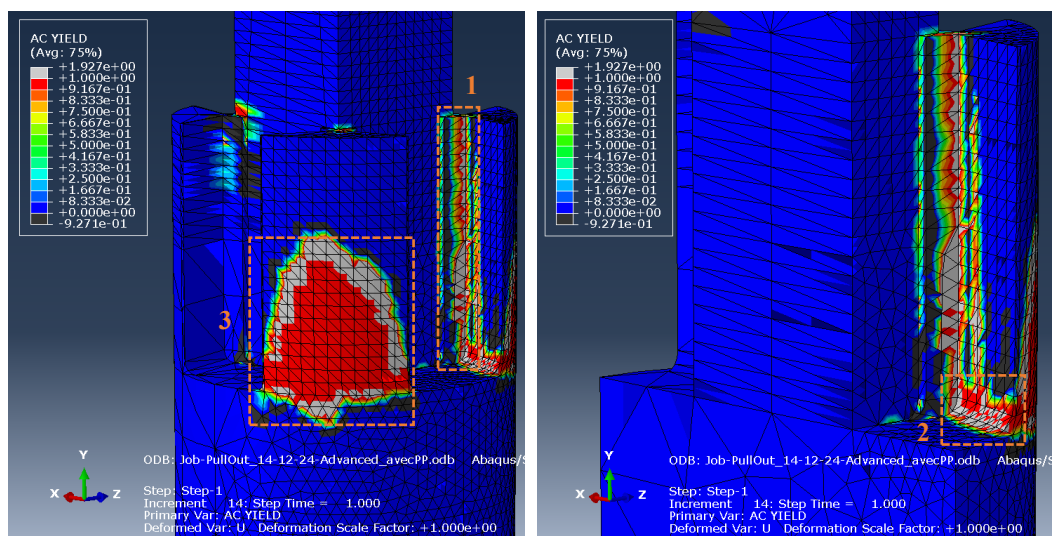


Figure 82: In red and grey, finite elements which have entered plasticity in the column. Orientation of the views visible through the triad visible on the bottom left of the figures. The numbers and squares in orange highlight the fracture areas.

The Figure 83 highlights the displacement of the finite elements of the column in the z-axis (U3) in the contact area, at the end of the simulation. It is important to note that half the geometry was simulated and that the other half is visually added in Abaqus using a mirror. Therefore, the colours and values associated correspond to the values calculated for the right half geometry. The left half being a mirror of the right half. The displacements along the z-axis (U3) showcase a rotation of the vertical elements along the y-axis as the red signifies a displacement along the z-axis of approximately 7 [mm].

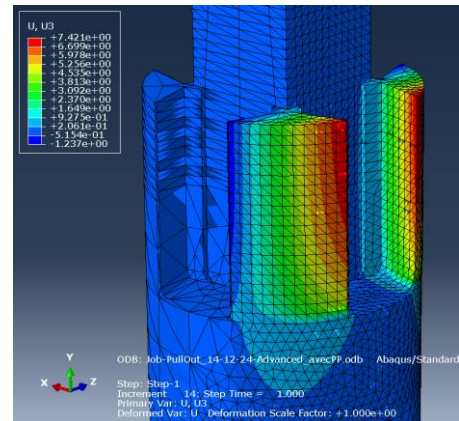


Figure 83: Displacement of the finite elements of the column in the z-axis, in the contact area, at the end of the simulation.

The Figure 84 highlights the stresses S11 of the finite elements of the column, in the contact area, at the end of the simulation. Stresses of approximately 7.5 [MPa] can be observed in the plasticized area 1.

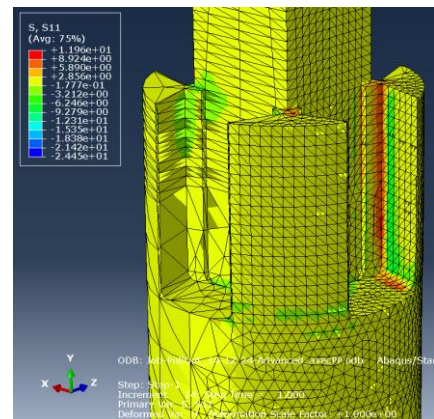


Figure 84: Stresses S11 of the finite elements of the column in the x-axis, in the contact

Afterwards, the Figure 85 highlights the finite elements of the beam which have entered in plasticity at the end of the test simulation. The area of plasticity forms a block on each side of the column corresponding to the area of the beam in contact with the column. This plasticity is mostly compression perpendicular to the fibres. The bottom part of the side of the dowel seems to be the area the most affected.

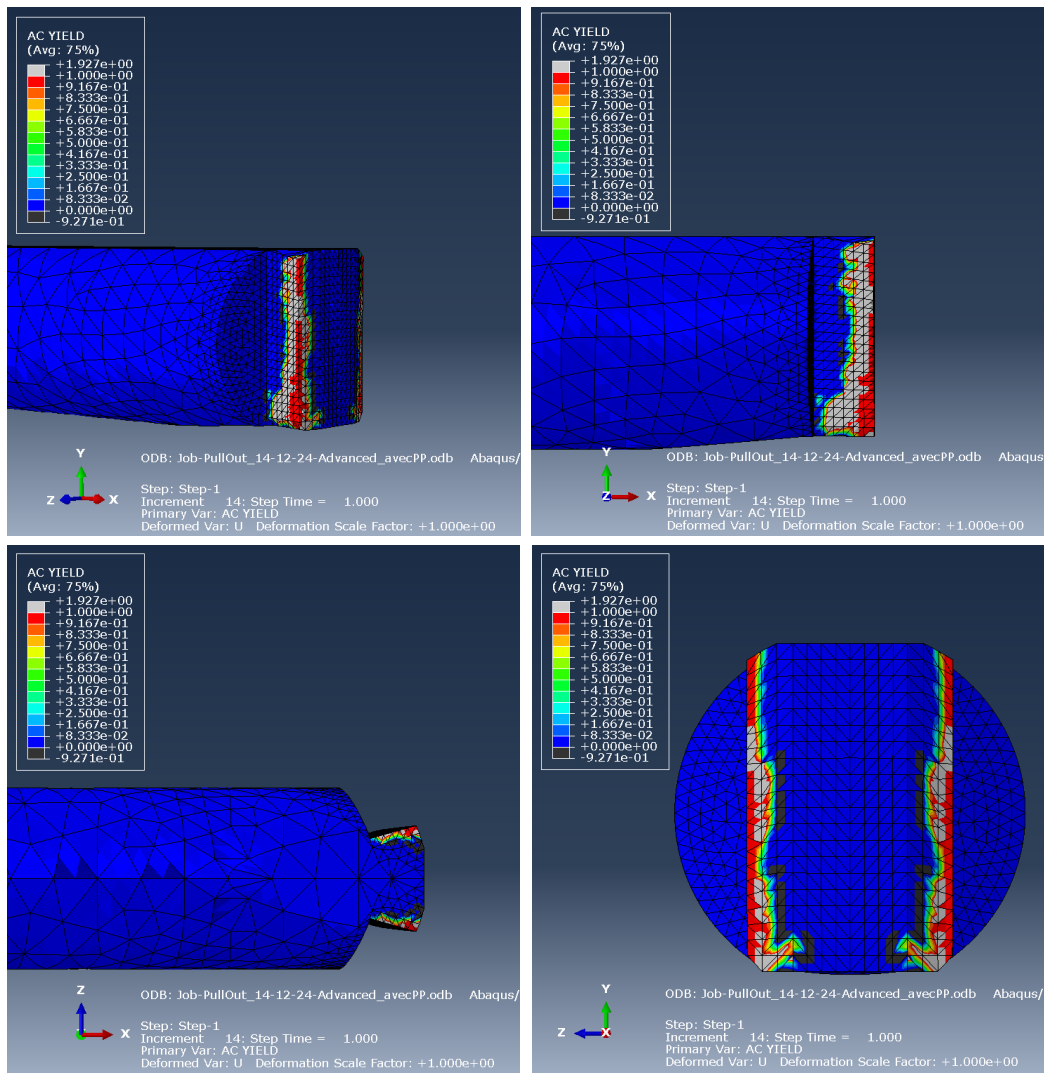


Figure 85: In red and grey, finite elements which have entered plasticity in the beam. Orientation of the views visible through the triad visible on the bottom left of the figures.

The Figure 86 highlights in blue the finite elements in the column which are in contact with the column at the end of the simulation. As we can see, the contact is concentrated on the sides and bottom of the tip of the beam.

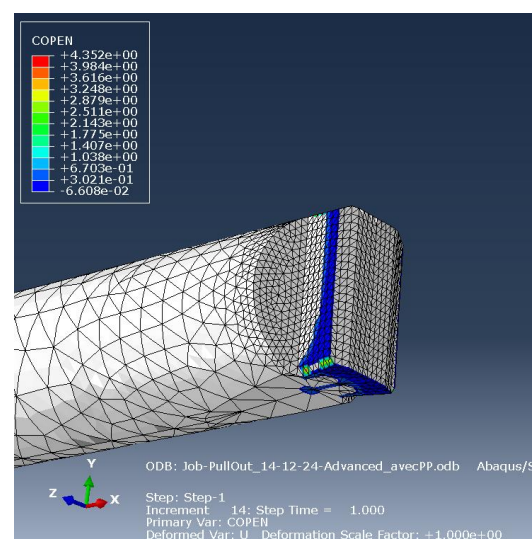


Figure 86: In blue: area of the beam in contact with the column, at the end of the test, after 20 [mm] of beam slip.

The Figure 87 showcases the displacements of the finite elements of the beam along the x-axis (U_1) at the end of the test simulation. Most of the beam is in blue as it displaced along the axis of the value of the actuator (20 [mm]). The elements in green, orange and red display the finite elements which are in contact with the column and therefore displace less than the other finite elements of the beam.

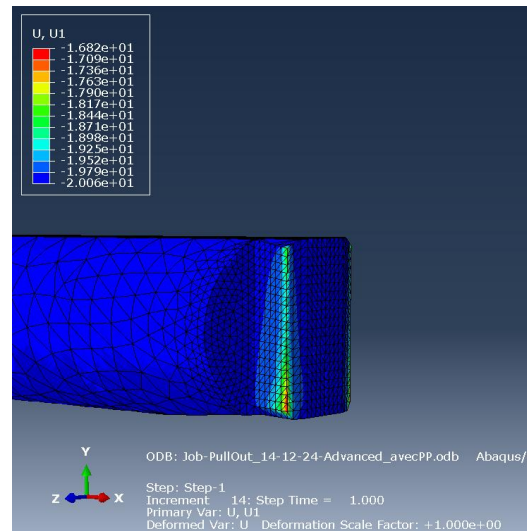


Figure 87: Displacements of the finite elements along the x-axis (U_1) for the beam, in the contact area, at the end of the simulation.

3.3. Comparisons between the experimental and numerical results

Key aspects of the section: In this section, the experimental and numerical results are summarized and compared to evaluate the strengths and weaknesses of the numerical model. The overall force-displacement graph shapes are similar, but tolerance accuracy varies between the experiment and simulation. The first cracks appear in both cases, though the second cracking location differs. Both approaches reveal three phases: elastic behaviour, plastic behaviour with strength reduction, and a plateau. However, the numerical actuator requires roughly double the force of the experimental one. Discrepancies may stem from the lack of calibration of elastic and plastic parameters and imprecise milling. Improved accuracy could be achieved through 3D scanning to match experimental geometries.

Summary of the experiment

Figure 88 shows the force-displacement graph at the actuator, revealing four distinct phases of the experiment:

- **Phase 1** (0–3.5 [mm]): This phase represents the structural strength of the assembly until cracks form in the vertical elements.
- **Phase 2** (3.5–10 [mm]): During this stage, the assembly primarily exhibits elastic behaviour, with its strength provided by the portion of the column located below the support level.

- **Phase 3** (10–15 [mm]): In this phase, the assembly continues to bear load despite cracks in the lower portion of the column. At 15 mm, the lower section of the column ruptures, transitioning into the final stage.
- **Phase 4** (15–22 [mm]): This final phase, ending at 22 mm (for comparison with the numerical test), is characterized by reduced strength and significant plastic deformation of the assembly.

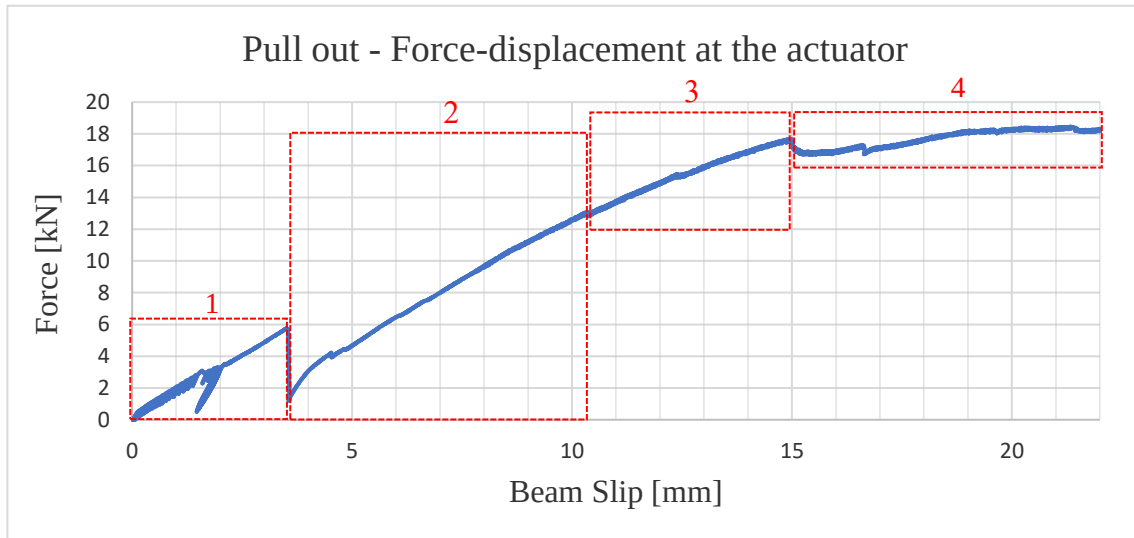


Figure 88: Force-displacement curve experimentally measured at the pulling actuator.

Summary of the numerical simulation

Figure 89 illustrates the force-displacement graph for the actuator, as obtained from the numerical simulation. The graph is divided into four distinct phases:

- **Phase 1** (0–3.27 [mm]): In this initial stage, the beam slides against the column with friction until contact occurs between the vertical elements of the column and the sides of the beam.
- **Phase 2** (3.27–10 [mm]): This phase captures the elastic response of the assembly, during which the system resists deformation until the first finite elements in the vertical components begin to yield.
- **Phase 3** (10–16 [mm]): During this phase, both the column and beam exhibit plastic behaviour. The strength of the assembly is sustained as plasticity develops further, as described in 3.2.
- **Phase 4** (16–20 [mm]): In the final phase, the assembly is heavily plasticized, resulting in a slight softening of its structural response.

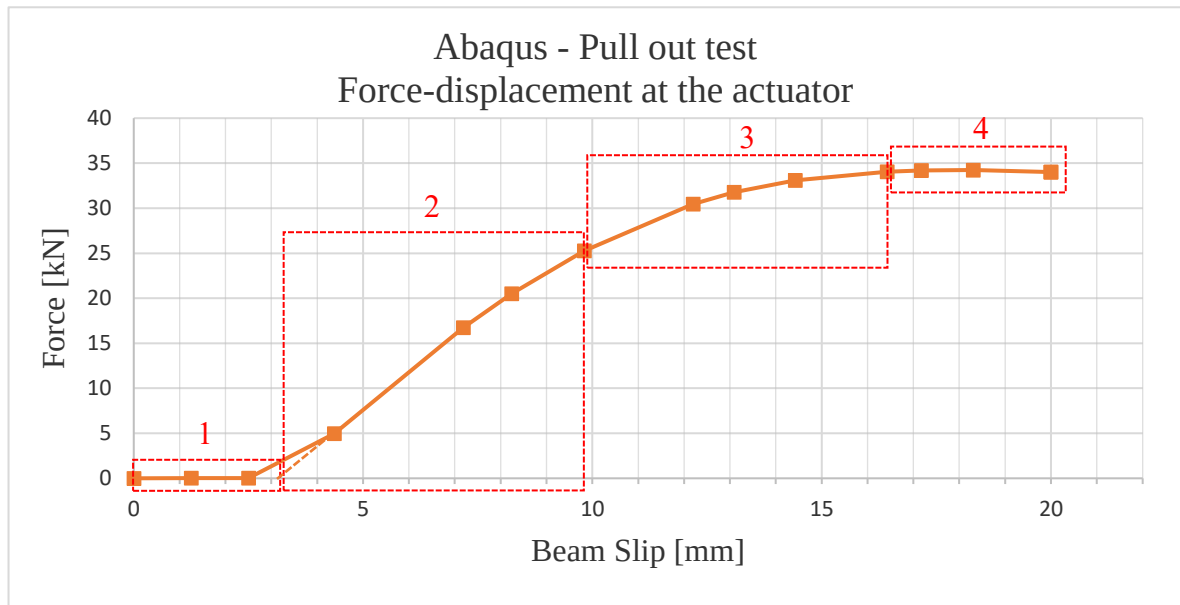


Figure 89: Force-displacement curve measured at the pulling actuator in Abaqus.

Comparison between the experimental and numerical results

The experiment and the numerical results differ when it comes to the beginning of the test. The experiment does not show a slippage required for the beam and column to enter in contact as they are in contact before the experiment starts. This is due to the impossibility of respecting the required tolerances experimentally whereas the numerical simulation, with perfectly plane surfaces and exact positioning, showcases a horizontal slip of the beam with little friction resistance up till the contact appears between the sides of the tip of the beam and the vertical elements of the column.

Taking that into account, the first part of the numerical simulation can be omitted to compare the numerical simulation to the experiment starting from the moment of contact between the vertical elements of the column and sides of the beam. It is interesting to note that the overall shape of the force-displacement graph is similar between the experiment and the numerical test. Moreover, the cracks observed experimentally in the vertical elements of the column are also observed numerically (through plastification of finite elements) but are visible at a difference load of the actuator and different beam slip.

The crack happening experimentally after 3.5 [mm] of beam slip, or 6 [kN] of force at the actuator, appears after approximately 6.5 [mm] (additional slippage since contact being deduced) in the numerical simulation and 25 [kN] of force at the actuator. The behaviour of the assembly observed in the part two and three of the experiment is also observed in the part three of the numerical simulation as the rigidity of the assembly reduces the more the cracks propagate.

The crack seen experimentally in the lower part of the column and visible in the Figure 71 is not found in the numerical simulation. Instead, a bulb of finite elements in plastification is seen a bit higher, as visible in the Figure 82 and highlighted by an orange square and a number '3'. This difference between the numerical model and the experiment is probably due to the way the first rupture is modelled. Due to the current modelization of the non-linearities, the first rupture, in the numerical simulation, is modelled by a strong deformation of the material with a reduced strength between the fibres. Whereas, experimentally, a complete loss of strength is observed in this cracked area. This is a limit of the current numerical model and it is a key area which could benefit from improvements. Other improvements could also be made regarding the elastic properties of the material which would benefit from a small test campaign on the exact type of wood aimed for the experiment and to be simulated numerically. Lastly, the plastic behaviour observed experimentally after 15 [mm] of beam slip is well modelled numerically.

Other parameters that could explain the difference in strength required to pull on the beam for the experiment compared to the numerical simulation could be the dimensions of the assembly used for the experiment. In fact, at the location where the first cracks appear experimentally (at the vertical elements of the column), a theoretical width of 22.4 [mm] is expected whereas we can see from the Figure 90 that this dimension was not respected experimentally. The assembly's dimensions are not as accurate as initially expected therefore highly influencing the results of the test.

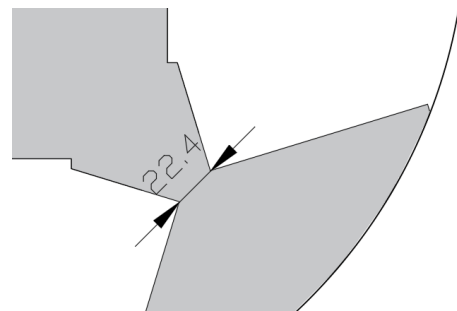


Figure 90: Theoretical local dimension of the vertical elements in the column.

In conclusion, it is pertinent for the future research to keep improving the numerical model regarding the modelization of fractures as well as the elastic and plastic parameters to corroborate better with the tested wood. A calibration of the numerical model could be done by a small experimental campaign on small clear wood samples of the wood to be tested. Once done, then a broader variety of complex tests like the pull-out test here described could take place and be compared to the latest numerical model developed. The quality of the wood and the accuracy of the dimensions of the timber elements need to be higher than what was here tested. Moreover, it would be interesting to scan the exact geometry tested experimentally and import this geometry in Abaqus for the numerical simulation in order to test on a model closer to reality and therefore increasing the accuracy of the numerical results.

Page intentionally left blank for printing purposes.

Chapter 4

Conclusion

4. Conclusion

Multi-storey buildings are crucial in our modern society, from residences and offices to educational institutions. However, the construction sector remains a significant contributor to climate change. Fortunately, wood has emerged as a promising material due to its favourable environmental properties. Compared to traditional materials like steel and concrete, wood is highly competitive from a sustainability perspective.

Despite these advantages, the widespread adoption of wood faces a major obstacle: its cost, which is currently higher than its conventional rivals. To address this, a collaborative project was initiated, bringing together three companies (MOBIC, DUCHÊNE, and Buildwise), a university (UCLouvain) and governmental support from SPW – Service Public de Wallonie. The project aims to develop technical solutions for constructing multi-storey buildings using circular debarked logs. The goal is to create a product that combines exceptional environmental performance with cost-competitiveness, making wood a viable and sustainable choice for construction. One promising approach explored by the team involves designing wood-only assemblies fabricated using robotic milling.

This approach necessitates the design and evaluation of assemblies, through experiments, to assess their strength, load behaviour and other mechanical properties. While these experiments are critical, this master's thesis focused on the numerical modelling of such assemblies to simulate real-world phenomena. Numerical modelling offers several advantages such as reducing the need for extensive physical testing and providing deeper insights into material behaviour.

Currently, no comprehensive user-friendly wood material model exists that can be readily applied in numerical simulations. This limitation arises from the complexity and variability of wood which differs significantly across species and requires specific parameters obtained from tests or databases. To address this gap, this thesis presented a step-by-step guide for implementing a simplified numerical model for wood in Abaqus. Key modelling decisions were outlined and critically analysed.

It is important to note that this thesis does not claim to have developed a perfectly accurate wood model as further research is still necessary. However, the results are encouraging, offering several potential benefits: enhanced understanding of wood's behaviour under load, the ability to simulate numerous test scenarios more efficiently and cost-effectively, improved building safety and optimized material usage. These advantages highlight the potential for wood to be modeled numerically.

The work is organized into three main chapters, each addressing a critical aspect of the study.

The first chapter introduced the central problem, emphasizing the importance of assemblies in building construction due to their impact on safety, cost, and durability. This significance is even more pronounced when wood is used, given its complex properties. The chapter also explored the environmental impact of the construction industry, framing the original problem and positioning wood as a sustainable alternative—provided its harvesting is conducted responsibly. The chapter concluded by outlining the intricate properties of wood, some of which are modelled numerically in the subsequent work.

The second chapter, titled ‘Numerical Simulations of Wood’, began with a concise review of the current state of numerical analysis of wood. It identified that while several advanced numerical models exist, their high complexity often makes them inaccessible to engineers or researchers without advanced expertise in numerical modelling. To address this gap, the study proposed a simplified numerical model of wood designed to balance accuracy and usability. The steps for developing this model are thoroughly described and critically analysed to ensure reproducibility. The chapter also introduced the Finite Element Method (FEM) and the FEM software used in this study, Abaqus.

The second part of this chapter explored different material models for wood, ranging from a basic yet inaccurate elastic model to more advanced orthotropic models for rectangular and circular cross-sections. These models were evaluated through introductory simulations, including a simply supported beam and an overhanging beam. The results demonstrated that the simple isotropic model consistently deviates the most from analytical solutions, whereas the tailored isotropic model for wood and the orthotropic model yielded results closer to analytical predictions. The chapter concluded with a discussion of critical considerations in wood modelling, such as contact definitions, tolerances and non-linearities.

The third chapter focused on an experimental pull-out test of a wood-wood column-beam dovetail assembly used to confront the final numerical model. This chapter began with a detailed description of the experimental setup and results, including a force-displacement graph from the pulling actuator and photographic documentation of the outcomes. It then described the numerical simulation replicating the experiment and the two scenarios were compared. Despite its simplicity, the numerical model captured key fracture modes and reasonably approximated the progressive loss of strength and final failure. However, limitations include its inability to accurately simulate crack behaviour under tension and the need for calibrated elastic and plastic parameters tailored to the tested wood. The chapter concluded by discussing potential future improvements to enhance the model’s accuracy and practical applicability.

Recommendations for Future Research

The final numerical model of wood developed in this study has a notable limitation: its inability to accurately simulate fractures that occur in wood under tension, either perpendicular or parallel to the fibres. Addressing this issue would significantly enhance the model's accuracy.

Furthermore, the elastic and plastic parameters used in the model were sourced from the literature and correspond to the same wood species tested experimentally. However, conducting a small experimental campaign on test samples of the exact wood type to be used in the experiments or the building would allow for precise property extraction and model calibration, further improving accuracy.

Improved milling precision for wood-wood assemblies is also essential, not only for experimental testing but for the reliable use of such assemblies in future buildings. For comparative purposes, 3D scanning the experimental elements and integrating the resulting geometries into the numerical model would reduce discrepancies between experimental and numerical results.

Continued development of the numerical model requires additional experiments and their numerical reproduction to evaluate the phenomena captured by the model and identify areas for improvement. Recommended tests include pull-out tests and moment-rotation tests, which are particularly relevant for understanding wood assembly behaviour.

Finally, to further improve the numerical model and the understanding of the material wood, extraction of more experimental data would be pertinent. Suggested data points include results from digital image correlation (DIC), displacement measurements at multiple locations using optic fibres and other detailed observations that provide a richer basis for comparison with the numerical model.

Annexes

5. Annexes

5.1. Most common failure criteria

A failure criterion is a rule or mathematical model used to predict the failure of a material under loading. The criterion usually defines the transition between the elastic domain of the material and its plastic domain, or fracture. The most common failure criteria used for wood in the literature are here briefly described. The implementation of a failure criterion requires the use of a user subroutine (UMAT/VUMAT), or a plug-in, which is a complex task. A simpler way to identify the yielding of the material, or its fracture, is to determine the yield stress ratios as used in this master's thesis and described in Section 2.5.3.

1. **Hankinson's criterion:** one of the most broadly used criterion for compression and elastic limit at a varying angle with respect to fibres direction. It is currently used in the EC5. It can be summarised as follow (See Equation 7):

$$S_{\theta} = \frac{S_{0^{\circ}} \cdot S_{90^{\circ}}}{S_{0^{\circ}} \cdot \sin^2(\theta) + S_{90^{\circ}} \cdot \cos^2(\theta)}$$

Equation 7: Hankinson's criterion for load at a varying angle.

With:

- S_{θ} : strength of wood at an angle θ to the fibres.
 - $S_{0^{\circ}}$: strength of wood parallel to the fibres.
 - $S_{90^{\circ}}$: strength of wood perpendicular to the fibres.
2. **Von Mises' criterion:** commonly used to predict the start of plastic deformation for ductile materials under complex loading conditions. Yielding is considered to start when the distortional component of the stress reaches a certain threshold.

$$\sigma_v = \sqrt{\frac{1}{2} [(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2]}$$

Equation 8: Von Mises' criterion.

With:

- $\sigma_1, \sigma_2, \sigma_3$: principal stresses.
- σ_v : Von Mises equivalent stress, to compare to material's yield strength.

If the Von Mises equivalent stress exceeds the yield strength, plastic deformation occurs. This criterion is usually used for isotropic materials but can be adapted. It is generally not recommended for wood.

3. **Hill's yield criterion:** one of the most commonly used for anisotropic materials, it can be considered an extension of the Von Mises' criterion to account for anisotropy in materials.

$$F(\sigma_2 - \sigma_3)^2 + G(\sigma_3 - \sigma_1)^2 + H(\sigma_1 - \sigma_2)^2 + 2L\tau_{23}^2 + 2M\tau_{31}^2 + 2N\tau_{12}^2 = 1$$

Equation 9: Hill's yield criterion.

With:

- $\sigma_1, \sigma_2, \sigma_3$: principal stresses.
 - $\tau_{12}, \tau_{23}, \tau_{31}$: shear stresses.
 - F, G, H, L, M, N: material constants derived from yield stress ratios in the different directions.
4. **Tsai-Wu's criterion:** very commonly used as well to describe elastic-brittle materials and is often found to be conservative. The criterion is a polynomial failure theory predicting failure based on a combination of stress states. It has the strong advantage to be able to take into account the anisotropy of wood.

$$F_1\sigma_1 + F_2\sigma_2 + F_{11}\sigma_1^2 + F_{22}\sigma_2^2 + 2F_{12}\sigma_1\sigma_2 + F_{66}\tau_{12}^2 \leq 1$$

Equation 10: Tsai-Wu criterion.

With:

- σ_1, σ_2 : normal stresses in the principal material's directions.
- τ_{12} : shear stress in the plane of the material.
- $F_1, F_2, F_{11}, F_{22}, F_{12}, F_{66}$: material-dependent coefficients related to the strength properties of the material. Constants being derived from uniaxial tension, compression and shear strength tests.

However, the criterion holds two limitations. The first one is regarding the failure mode, it cannot directly predict the type of failure occurring. Secondly, detailed material testing is needed to determine the various strength parameters needed for the criterion.

5. **Norris' criterion:** also made for orthotropic materials in the first place, it can predict failures under combined stress states.

$$\frac{\sigma_1^2}{X_t X_c} + \frac{\sigma_2^2}{Y_t Y_c} + \frac{\tau_{12}^2}{S^2} \leq 1$$

Equation 11: Norris' criterion.

With:

- σ_1 : normal stresses in the longitudinal direction, parallel to the fibres.
- σ_2 : normal stresses in the transverse direction, perpendicular to the fibres.
- τ_{12} : shear stress in the plane of the material.
- X_t, X_c : tensile and compressive strengths in the longitudinal direction.
- Y_t, Y_c : tensile and compressive strengths in the transverse direction.
- S : shear strength in the plane of the material.

Compared to Tsai-Wu criterion, Norris is a bit simpler as it does not include an interaction between the normal stresses and the shear. It makes it easier to apply but potentially less accurate if the loading is complex.

6. **Hoffman's criterion:** Similar to Tsai-Wu criterion, it also aims to predict failures of orthotropic materials.

$$F_1\sigma_1 + F_2\sigma_2 + F_3\sigma_3 + F_{11}\sigma_1^2 + F_{22}\sigma_2^2 + F_{33}\sigma_3^2 + 2F_{12}\sigma_1\sigma_2 + 2F_{13}\sigma_1\sigma_3 + 2F_{23}\sigma_2\sigma_3 \leq 1$$

Equation 12: Hoffman's criterion.

With:

- $\sigma_1, \sigma_2, \sigma_3$: normal stresses in the principal material's directions.
 - F_1, F_2, F_3 : linear strength coefficients.
 - F_{11}, F_{22}, F_{33} : quadratic strength coefficients.
 - F_{12}, F_{13}, F_{23} : interaction strength coefficients between normal stresses.
7. **Hashin damage model:** failure criterion initially developed for fibre-reinforced composites. It also works very well for laminated composite materials where the strength in the different directions is dependent on fibres and a matrix. Like previously mentioned criteria, it aims at predicting the onset of failure. Additionally, it aims at simulating how damage evolves after the failure starts. This makes it more accurate but also more complex. In fact, it uses different criteria to predict different types of failures: Fibre tension failure, Fibre compression failure, Matrix tension failure and matrix compression failure. All having their own failure equation.
 8. **Quadratic Mean Stress (QMS) criterion:** it expresses the failure condition in terms of a quadratic combination of the principal stresses acting on the material. Stresses in different directions are related to a failure surface. It is relatively simple and therefore may not capture all failure modes of wood such as fibre crushing or local buckling.

$$\sigma_1^2 + \sigma_2^2 + \sigma_3^2 \leq \sigma_{crit}^2$$

Equation 13: Quadratic Mean Stress criterion.

With:

- $\sigma_1, \sigma_2, \sigma_3$: normal stresses in the principal material's directions.
- σ_{crit} : critical stress defining the material's failure threshold linked to its mechanical strength.

5.2. Determination of the orthotropic parameters

The elastic parameters mentioned in the section 2.3 Materials models can be obtained by performing six distinct tests, on clear wood specimen, as described in the Figure 91.

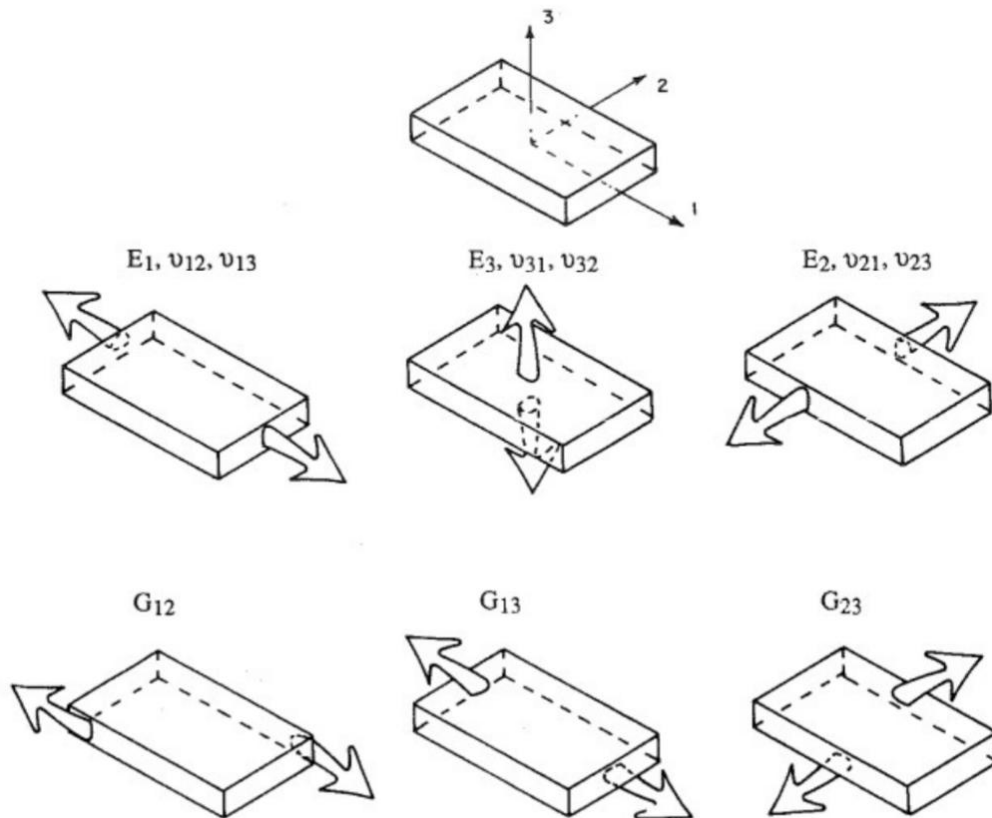


Figure 91: "Illustration of the traction and shearing tests necessary for the determination of the engineering constants of an orthotropic material". Source : LMECA2640- Mechanics of Composite Materials, Issam Doghri (2022)

5.3. Mesh choice and mesh convergence study

For this master's thesis, it was decided to use linear tetrahedral finite elements due to their ability to fit easily any geometry together with their efficiency as the computational time needed to be reasonable with the computers available.

In Section 2.4.1, an introductory exercise is performed to analyse the impact of the choice of material model on results like stresses and displacements. The first exercise consists of a simply supported beam with a surface load on its top surface. To assess what density of finite elements would be suitable for this exercise, a mesh convergence study is performed. It consists in running the same simulations several times with a varying density of finite elements and extract here two key results: the maximum stress and the deflection at mid-span of the beam. As a reminder, **linear tetrahedral** FE are here used and the material model is **isotropic for wood**.

Here, the density of the mesh is analysed based on a parameter that is implemented in Abaqus. This parameter is called the “approximate global seeding size”. This parameter refers to non-strict constraint that the user specifies to Abaqus regarding the distance between seeds²⁵ of the mesh. Based on this value, Abaqus will try to create a mesh. This constraint is non-strict as Abaqus might decide that a finer mesh is necessary, for example to fit-in complex geometries. This applies to the entire model unless it is overridden by local seeding or mesh controls. The value entered in Abaqus uses the same units as the model’s dimensions. Namely, millimetres here.

Mesh convergence study for the squared section

As visible in the Table 25, the Figure 92 and the Figure 93, the accuracy of the numerical results (in blue) is much better when the global seeding size is between 20 and 10 [mm]. Interestingly, the maximum stress and displacement observed at mid-span for an approximate global seeding size of 10 (7.54 [MPa] & 12.52 [mm]) is less close to the analytical solution (7.50 [MPa] & 12.49 [mm]) than for an approximate global seeding size of 15 (7.51 [MPa] & 12.49 [mm]). This could be due to the numerical model having slightly different hypotheses than the analytical model or numerical irregularities. It could be interesting to continue refining the mesh to see the impact on the results however, the student version here used did not allow for a finer mesh. Luckily, it can be seen that the results are not drastically changing from a global seeding size of 15 to 10 [mm] informing us that a seeding size of 15 or 10 [mm] is appropriate for reasonably accurate results.

²⁵ In Abaqus, seeds and nodes serve distinct roles in the finite element process. **Seeds** are pre-meshing tools used to define the size and distribution of elements along a geometry, guiding how the mesh is generated. In contrast, **nodes** are points within the resulting mesh where the analysis occurs, providing the degrees of freedom for computations such as displacement or stress. While seeds determine where nodes will be placed, nodes are integral to the finite element model and persist throughout the analysis.

Approximate global seeding size [mm]	σ [MPa]	δ [mm]
80	6.00	11.43
60	6.47	11.58
40	7.04	12.17
20	7.47	12.44
15	7.51	12.49
10	7.54	12.52
Analytical model	7.50	12.49

Table 24: Results of the mesh convergence study of the squared section of the exercise 1.

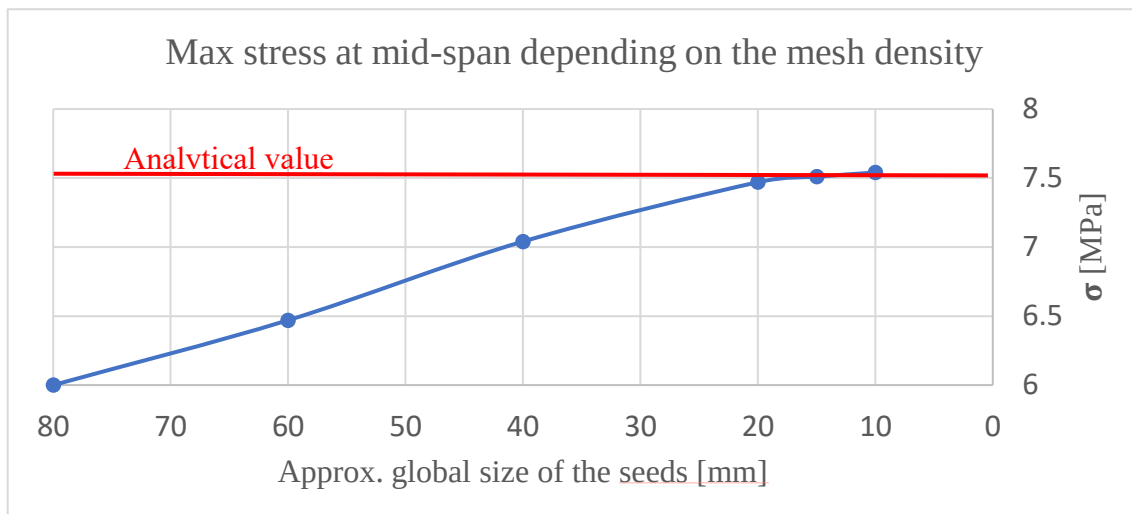


Figure 92: Graph of the maximum stress obtained at mid-span of the beam of the exercise 1 for a squared section for varying approximate global sizes of the seeds.

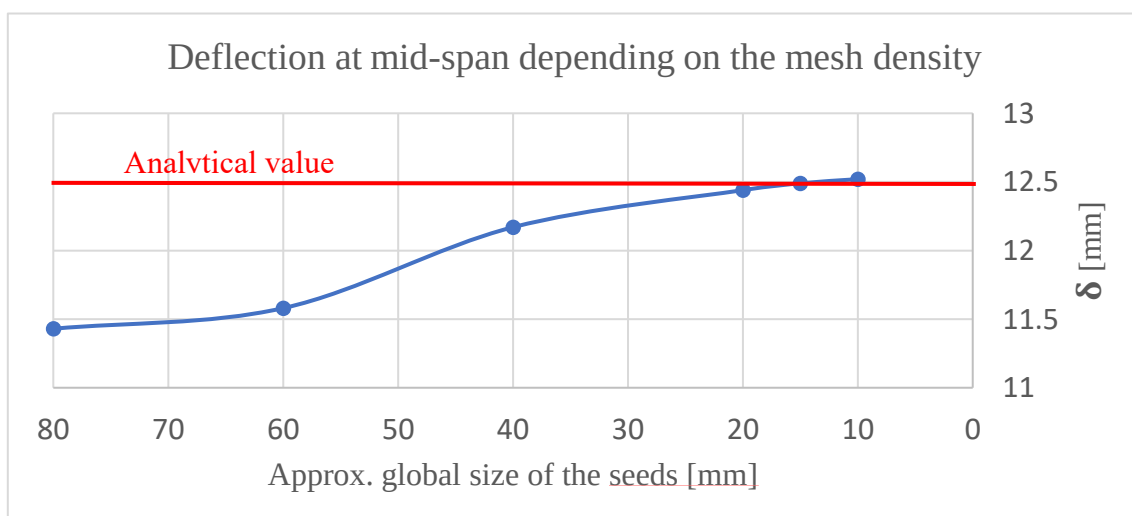


Figure 93: Graph of the deflection obtained at mid-span of the beam of the exercise 1 for a squared section for varying approximate global sizes of the seeds.

Mesh convergence study for the circular section

Similarly to the rectangular section, the circular section demonstrates that a global seeding size of 20 to 10 [mm] is rather accurate (see Table 25, Figure 94, Figure 95). In this case however, with the finest mesh here modelled (with global seeding size of 10), the stress obtained numerically is exactly the same as analytically but the numerical displacement at mid-span (12.53 [mm]) is still slightly inferior to the analytical result (12.64 [mm]). This could be due to differences in hypotheses between the numerical model and the analytical model.

Approximate global seeding size [mm]	σ [MPa]	δ [mm]
80	5.30	11.04
60	5.70	11.51
40	6.05	12.16
20	6.42	12.45
15	6.44	12.50
10	6.49	12.53
Analytical model	6.49	12.64

Table 25: Results of the mesh convergence study of the circular section of the exercise 1.

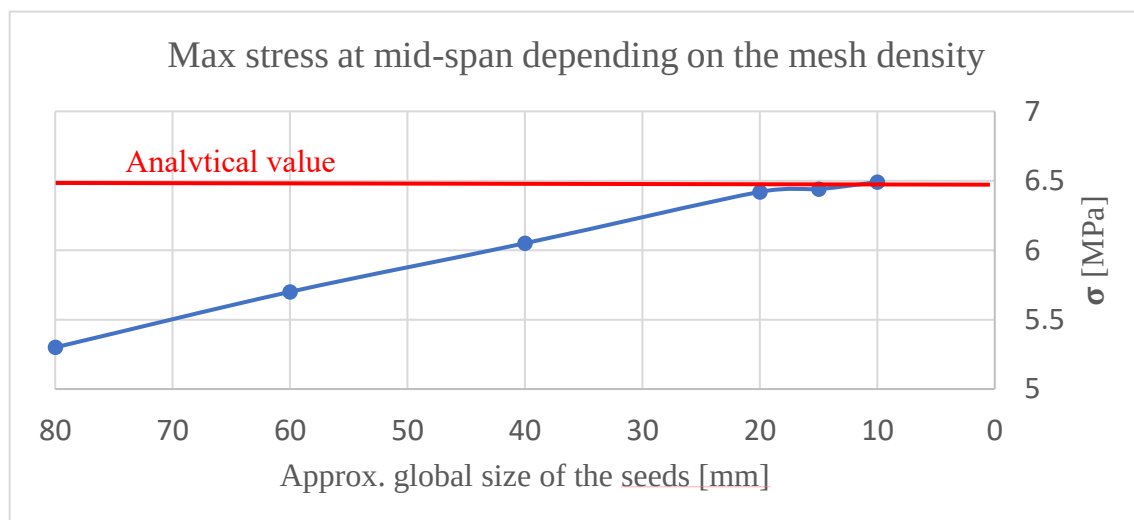


Figure 94: Graph of the maximum stress obtained at mid-span of the beam of the exercise 1 for a squared section for varying approximate global sizes of the seeds.

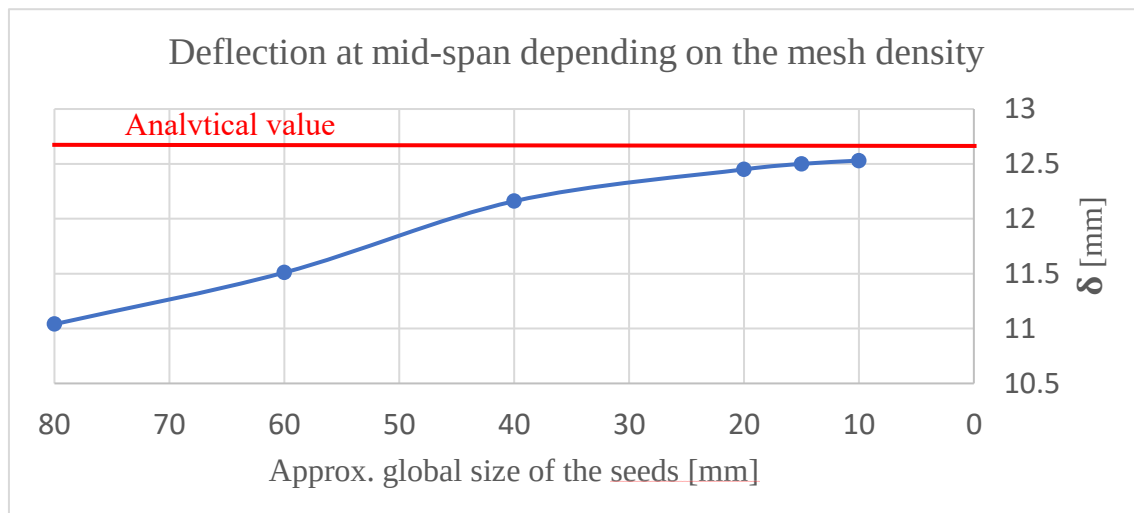


Figure 95: Graph of the deflection obtained at mid-span of the beam of the exercise 1 for a squared section for varying approximate global sizes of the seeds.

From this mesh convergence study, it was decided for the introductory exercises one and two to use linear tetrahedral FE with a global seeding size of 10 [mm]. For the rest of the master's thesis, the same type of finite element was used and the mesh density (through the approximate global seeding size) was chosen to respect a compromise of efficiency and accuracy together with the limit of 250'000 nodes allowed by the educative version of Abaqus available.

5.4. Tutorial – key steps in Abaqus CAE

As the steps in Abaqus can quickly be overwhelming, a small tutorial is here presented. This tutorial gathers the main steps to follow to model wood in Abaqus. This tutorial is not intended to be detailed but only an introduction to the steps to follow for the reader to have a global idea and possibly reproduce it. The reader still bears entire responsibility to take the time to understand the different steps and choices possible and their consequences on the results of a numerical analysis.

The numerical analysis here presented requires 10 steps, described briefly and sometimes illustrated. For more detailed tutorials, we recommend reaching out to the material presented in the course of “Numerical Analysis of Civil Engineering Structures” (39) as well as the various books and YouTube tutorials available.

Step 1. Modelization of the geometry of the elements

For this, the “Part” module must be selected, followed by “Create Part”. Alternatively, a geometry can be imported.

Step 2. Determination of the three main directions of the material

In order to model an orthotropic material model like described in the Section 2.3.

Select: Tools > Datum > CSYS – 3 points > Rectangular or Cylindrical, depending on the cross section of wood you are modelling. See Section 2.3 for more details on the subject.

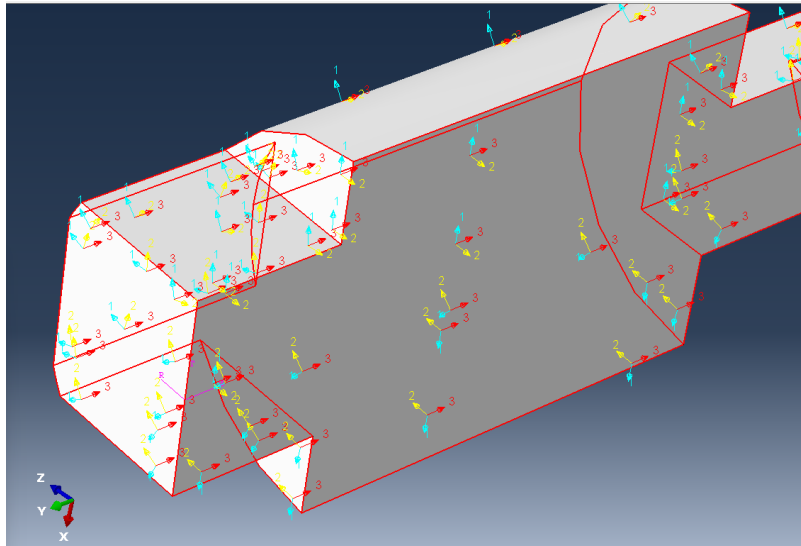


Figure 96: Example of result of the determination of the three main direction of wood. Here, an orthotropic material model is considered with 1, 2 and 3 corresponding to the radial, tangential and longitudinal axes, respectively.

Now, in the “Property” module:

Assign material orientation > part > CSYS > Datum CSYS list and choose the orthotropic model previously generated.

Step 3. Creation of the material

Still in the “Property” module: “Create Material”.

To model the elastic properties of the material: Mechanical > Elastic > Engineering constants. And enter the values of interests gathered from the literature or experimental tests. See Figure 97.

To model the plastic properties of the material: Mechanical > Plastic.

And enter the values of interests gathered from the literature or experimental tests. See Figure 98. For more information’s, reach for the Subsection 2.5.3 Non-linearities where the Yield Stress, Plastic Strain and the suboption “Potential” are detailed.

The properties made must now be linked to a section: Create section > Assign section.

Edit Material

Name: Material-wood

Description:

Material Behaviors

Elastic

General Mechanical Thermal Electrical/Magnetic Other

Elastic

Type: Engineering Constants

☐ Use temperature-dependent data

Number of field variables: 0

Moduli time scale (for viscoelasticity): Long-term

☐ No compression

☐ No tension

Data

	E1	E2	E3	Nu12	Nu13	Nu23	G12	G13	G23
1	770	565	11000	0.39	0.036	0.029	80	720	880

Figure 97: implementation of the elastic parameters in Abaqus.

Edit Material

Name: Material-wood

Description:

Material Behaviors

Elastic

Plastic

General Mechanical Thermal Electrical/Magnetic Other

Plastic

☐ Use scale stress value:

Hardening: Isotropic

☐ Use strain-rate-dependent data

☐ Use temperature-dependent data

Number of field variables: 0

Data

	Yield Stress	Plastic Strain
1	21	0
2	22	0.008

Suboption Editor

Potential

☐ Use temperature-dependent data

Number of field variables: 0

Data

	R11	R22	R33	R12	R13	R23
1	0.2	0.2	1	1	0.35	0.2

OK Can

Figure 98: Implementation of the plastic parameters in Abaqus.

Step 4. “Assembly” module

In the “Assembly” module: Create instance > Dependant (Mesh on part).

During this step, several parts can be brought as instances. Afterwards, they can be moved (with translational and rotational tools) in the space to be positioned exactly like required.

Step 5. “Step” module

In the “Step” module: Create step. This module allows for different loads to be applied at different moments. Generally, the boundary conditions will be applied from the start till the end but certain loads might need to be applied only during a certain step. This is also the

module allowing to determinate the number of increments per step. This parameter can be essential to have enough increments when the evolution of the consequences of a load is needed or to generate a graph with enough data.

Step 6. “Interactions” module

If there are multiple parts in the numerical model, the interactions between the parts must be specified in Abaqus. If not, no interaction will appear between the elements and the parts will simply ignore each other.

In the “Interaction” module, “Create contacts”. For more details, come back to Subsection 2.5.1. Contacts.

Step 7. “Load” module

Two main possibilities: “Create Load” and “Create Boundary Condition”.

The boundary conditions are usually applied from the initial step and the loads from the step 1. Many types of mechanical loads are available in Abaqus: concentrated force, pressure, line load, etc.

Step 8. “Mesh” module

For each element of the model, perform the following steps.

Object: select “Part” instead of “Assembly”.

Mesh > Controls > select the Element Shape (of the FE) desired.

Seed Part > Implement an “Approximate global size” of the seeds to have a compromise between accuracy and efficiency. Select “Mesh Part Instance”.

Step 9. “Job” module

Create job > Submit.

Step 10. “Visualization” module

In the “Visualization” module, the results of the numerical analysis can be observed: displacements, stresses, etc.

6. List of abbreviations

FSC	Forest Stewardship Council	An international organization that promotes responsible forest management by certifying forests and timber products that meet strict environmental, social and economic standards.
PEFC	Programme for the Endorsement of Forest Certification	A global alliance of national forest certification systems ensuring that forests are managed sustainably, promoting reforestation and conservation.
CNC	Computer Numerical Control	A technology that uses computers to automate and precisely control machine tools, enabling complex manufacturing processes.
FEM	Finite Element Method	A numerical technique for solving complex engineering problems by dividing structures into smaller, manageable parts (finite elements) and analysing their behaviour under various conditions.
LVDT	Linear Variable Displacement Transformer	A sensor used to measure linear displacement or position changes with high precision, often employed in structural or material testing.
DIC	Digital Image Correlation	A non-contact optical technique used to measure strain, displacement and deformation of materials or structures by analysing changes in digital images.

7. List of figures

Figure 1: Current way of building vs. the RoboTronc's alternative. (1) (3)	11
Figure 2: Comparison of actions to reduce our carbon footprint (on a yearly basis). (77).....	16
Figure 3: Number of extreme weather events globally, by year. (78).....	16
Figure 4: Observed widespread and substantial impacts and related losses and damages attributed to climate change. (15)	17
Figure 5: World population (Billions) projections by the UN (2022).....	17
Figure 6: Diagram of the chemical flows created by trees (17).....	18
Figure 7: Proportion and distribution of global forest cover by climatic domain in 2020. Source: FAO.....	20
Figure 8: Countries members of the PEFC. Source: PEFC.	23
Figure 9: World roundwood production comprising industrial roundwood and wood to burn. Source: FAO.....	24
Figure 10: Achievable products from timber. Source: P.H. Fleming. (68)	25
Figure 11: Shape of a log with its sections and its medium fibre. (3)	26
Figure 12: Internal structure of a tree. Source: adapted from Airi Illiste.....	27
Figure 13: Layers of a tree trunk. Image by brgfx on Freepik.....	27
Figure 14: Orthotropic axes of wood together with the 3 main sections.	29
Figure 15: Scheme of the FEM for structural analysis. (33).....	36
Figure 16: The three fractural modes. Source: fracturemechanics.org.	39
Figure 17: Comparison of experimental results using ARAMIS system with Numerical results using Abaqus. (35)	40
Figure 18: Numerical simulations on wood's microstructure. (36)	43
Figure 19: Dog-bone geometry. (38)	45
Figure 20: Embedment test. (38)	46
Figure 21: Result of a Finite Element Method simulation on a bridge. (39)	48
Figure 22: Types of Finite Elements. (8)	48
Figure 23: Summary of Numerical Analysis of a structure. (39).....	50
Figure 24: On the left, Hourglass phenomenon. On the right, the correct result. (39)	51
Figure 25: Key steps of Abaqus.	53

Figure 26: The sections considered in Abaqus.	55
Figure 27: Introductory exercise 1.....	56
Figure 28: Wood beam (brown) and steel support (grey).	57
Figure 29: Boundary conditions in Abaqus.	57
Figure 30: cross-section of the rectangular wood beam with its steel boundary condition.	57
Figure 31: Lateral view of the stresses in the squared beam when considering an Isotropic for wood material model. Deformation scale factor: 20.....	60
Figure 32: Top view of the stresses in the squared beam when considering an isotropic for wood material model. Deformation scale factor: 20.....	60
Figure 33: Top cut and bottom support.....	61
Figure 34: Boundary conditions in Abaqus.	61
Figure 35: cross-section of the circular wood beam with its steel boundary condition.....	61
Figure 36: Lateral view of the stresses in the circular beam when considering an isotropic for wood material model. Deformation scale factor: 20.....	63
Figure 37: Top view of the stresses in the circular beam when considering an isotropic for wood material model. Deformation scale factor: 20.....	64
Figure 38: Introductory exercise 2.....	65
Figure 39: Top view of the stresses in the squared beam when considering an isotropic for wood material model. Deformation scale factor: 20.....	67
Figure 40: Lateral view of the stresses in the squared beam when considering an isotropic for wood material model. Deformation scale factor: 20.....	67
Figure 41: Top view of the stresses in the circular beam when considering an isotropic for wood material model. Deformation scale factor: 20.....	68
Figure 42: Lateral view of the stresses in the circular beam when considering an isotropic for wood material model. Deformation scale factor: 20.....	69
Figure 43: Scheme of the exercise 3.	70
Figure 44: Tolerance of 50 [mm] between the beam and the column for the subsection “Contacts”.....	72
Figure 45: S33 stresses of the beam and column (test 1). Deformation scale factor: 100.....	74
Figure 46: Test 4 with a cohesive contact rule. Deformation scale factor: 100.....	75
Figure 47: Displacement along the axis of the beam (z axis) for the 6 tests performed. Deformation scale factor: 500.....	76

Figure 48: Tolerance of 0.1 [mm] between the beam and the column for the subsection “Tolerances”	78
Figure 49: Lateral view of the higher-density mesh in the contact area.	79
Figure 50: Stresses (S33) in the contact area. Deformation scale factor of 1.	80
Figure 51: Characteristic strength in the four main loading scenarios of wood. (17)	81
Figure 52: Main axes considered and the yielding strengths of the faces in the three mutually orthogonal directions. Adapted from (42).	83
Figure 53: Highlight of the finite elements which entered plasticity (in grey) (test a., Deformation scale factor: 30).	86
Figure 54: Stresses (S33) in the contact area. Left: test a. for the original load. Right: test b. with the load multiplied by 4. (Deformation scale factor: 1)	87
Figure 55: Elements which have entered plasticity (in grey) for test b. Left: clamped side of the beam. Centre: right side of the beam. Left: top of the column. (Deformation scale factor: 30)	87
Figure 56: Stresses (S33) in the column for the test b. (Deformation scale factor: 50)	88
Figure 57: Technical drawings of the column. (a) Lateral view (b) 3D perspective (c) A-A cross-section (d) 3D perspective zoom of the connection area (detail B). Source : adapted from Justin Goosse.	92
Figure 58: Technical drawings of the column. (a) Top view (b) Lateral view (c) Bottom view (d) Zoom on the connection area. Source : adapted from Justin Goosse.	93
Figure 59: 3D perspective of the beam.	94
Figure 60: Geometry of the column and beam tested experimentally and numerically. Left: experiments in laboratory (source: Serena Van Nimwegen). Right: 3D model in Abaqus.	94
Figure 61: Top view of the column-beam assembly (column in light grey, beam in dark grey) and the tolerances used in the experiments and the numerical simulations.	95
Figure 62: Zoom on the dovetail assembly. Left: experiment in laboratory when the fracture has happened (source: Serena Van Nimwegen). Right: 3D model.	95
Figure 63: Lateral view of the 3D model with its boundary conditions.	96
Figure 64: (left) View of the cameras required for the DIC. (right) View of the location of the LVDT's on the column (same on each side of the column).	97
Figure 65: Views of the cracks observed on the column after the test. Source: Serena Van Nimwegen.	98
Figure 66: Views of the cracks observed on the column. Source : Serena Van Nimwegen. ...	99

Figure 67: loading procedure described in the ISO6891. (44).....	100
Figure 68: Zoom on the theoretical tolerances of the assembly column-beam.	100
Figure 69: Force-displacement curve measured at the pulling actuator.	101
Figure 70: First area with cracks observed experimentally.	102
Figure 71: Area of the column resisting to the beam pulling once the left vertical element has ruptured.	103
Figure 72: Area of the column cracking at approximately 15 [mm] beam slip.	103
Figure 73: 3D perspective of the geometry tested numerically in Abaqus.	105
Figure 74: Boundary conditions considered numerically.	106
Figure 75: View of the mesh of the beam before local seeding. In red, lines where a local seeding is implemented for the beam.....	107
Figure 76: In pink, lines where a local seeding is implemented for the column.	107
Figure 77: Final mesh on the beam.....	107
Figure 78: Final mesh on the column, in the contact area.	107
Figure 79: Top view of the dovetail assembly. Left: the initial position. Right: final position after 20 [mm] of beam slip.....	108
Figure 80: Force-displacement curve measured at the pulling actuator in Abaqus.	108
Figure 81: In red and grey, finite elements which have entered plasticity. Orientation of the views visible through the triad visible on the bottom left of the figures. The numbers and squares in orange highlight the fracture areas.....	110
Figure 82: In red and grey, finite elements which have entered plasticity in the column. Orientation of the views visible through the triad visible on the bottom left of the figures. The numbers and squares in orange highlight the fracture areas.	110
Figure 83: Displacement of the finite elements of the column in the z-axis, in the contact area, at the end of the simulation.	111
Figure 84: Stresses S11 of the finite elements of the column in the x-axis, in the contact area, at the end of the simulation.	111
Figure 85: In red and grey, finite elements which have entered plasticity in the beam. Orientation of the views visible through the triad visible on the bottom left of the figures. .	112
Figure 86: In blue: area of the beam in contact with the column, at the end of the test, after 20 [mm] of beam slip.....	112
Figure 87: Displacements of the finite elements along the x-axis (U1) for the beam, in the contact area, at the end of the simulation.....	113

Figure 88: Force-displacement curve experimentally measured at the pulling actuator.	114
Figure 89: Force-displacement curve measured at the pulling actuator in Abaqus.	115
Figure 90: Theoretical local dimension of the vertical elements in the column.	116
Figure 91: "Illustration of the traction and shearing tests necessary for the determination of the engineering constants of an orthotropic material". Source : LMECA2640- Mechanics of Composite Materials, Issam Doghri (2022).....	126
Figure 92: Graph of the maximum stress obtained at mid-span of the beam of the exercise 1 for a squared section for varying approximate global sizes of the seeds.	128
Figure 93: Graph of the deflection obtained at mid-span of the beam of the exercise 1 for a squared section for varying approximate global sizes of the seeds.	128
Figure 94: Graph of the maximum stress obtained at mid-span of the beam of the exercise 1 for a squared section for varying approximate global sizes of the seeds.	129
Figure 95: Graph of the deflection obtained at mid-span of the beam of the exercise 1 for a squared section for varying approximate global sizes of the seeds.	130
Figure 96: Example of result of the determination of the three main direction of wood. Here, an orthotropic material model is considered with 1, 2 and 3 corresponding to the radial, tangential and longitudinal axes, respectively.	131
Figure 97: implementation of the elastic parameters in Abaqus.....	132
Figure 98: Implementation of the plastic parameters in Abaqus.	132

8. List of tables

Table 1: Forest areas of the World. Source: FAO, 2021.	21
Table 2: Forest areas of Belgium. Source: Panorabois, 2021, OEWB.	21
Table 3: Top 10 Countries/Regions accounting for the most PEFC certified forests in the world. Source: PEFC.....	24
Table 4: Average shrinkage/swelling coefficient β of key species. (17)	30
Table 5: Thermal dilatation coefficient α of wood in its 3 main directions compared to steel and concrete. (17).....	31
Table 6: Elastic ratios of key species at approximately 12% water content. Adapted from (30).	32
Table 7: Poisson's ratios of key species at approximately 12% water content. Adapted from (30).	32
Table 8: Essential strength properties of key species at approx. 12% of water content. Adapted from (30).	33
Table 9: SI and consistent units chosen in Abaqus.	53
Table 10: Material models considered in Abaqus.	54
Table 11: Material parameters considered for the material models 3 and 4. (30).....	55
Table 12: Results obtained for the exercise 1.	59
Table 13: Results obtained for the exercise 2	66
Table 14: Main contact rules available in Abaqus.	71
Table 15: Results of the different contact rules.....	73
Table 16: Summary of the displacements of the beam in the direction parallel to its longitudinal axis at its right extremity, next to the contact, for each contact test performed.....	77
Table 17: Plastic rule considered in Abaqus.	82
Table 18: Yielding strengths in six key directions.	84
Table 19: Yield stress ratios considered in Abaqus.....	84
Table 20: Key results obtained in Abaqus.....	85
Table 21: Young moduli, Rigidity coefficients and Poisson ratios considered for the numerical pull-out test.	104
Table 22: Plastic rule considered in Abaqus.	104
Table 23: Yield stress ratios considered in Abaqus.	105

Table 24: Results of the mesh convergence study of the squared section of the exercise 1. .128

Table 25: Results of the mesh convergence study of the circular section of the exercise 1. .129

9. Bibliography

1. *Formulaire recherche - Appel à projet des pôles de compétitivité - Les troncs d'arbres pour la construction de bâtiments multi-étagés - RoboTronc*. Mobic, et al. 2019.
2. *EN1995: Eurocode 5 - Design of timber structures*. 2004.
3. Goosse, Justin. *Validation expérimentale d'assemblages de structures réciproques composées de troncs*. s.l. : UCLouvain, 2020.
4. Moens, Christopher and Niclaes, Jens. *Caractérisation mécanique des grumes*. Université catholique de Louvain : Ecole polytechnique de Louvain, 2021.
5. *Parametric design and robotic fabrication of whole timber reciprocal structures*. Geno, Julien, et al. 2022, Elsevier - Automation in Construction.
6. Dehaen, Maxime and Ponthiere, Pierre-Louis. *Recherche d'un protocole expérimental non destructif de caractérisation mécanique des grumes*. Université catholique de Louvain : Ecole polytechnique de Louvain, 2022.
7. Houben, Valentine. *Caractérisation mécanique non destructive des grumes à l'aide du GrindoSonic*. Université catholique de Louvain : Ecole polytechnique de Louvain, 2022.
8. Dupriez, Grégoire and Stevens, Ivan. *Numerical modelling of whole log wooden floors*. s.l. : UCLouvain - EPL, 2022.
9. Coster, Hugo De. *Recherche d'assemblages colonne-colonne-poutres pour bâtiments multi-étagés en bois rond*. s.l. : UCLouvain, 2023.
10. *Learning from failures in timber structures*. Frühwald, Eva. Lund Institute of Technology.
11. Frühwald, Eva, et al. *Design of safe timber structures – How can we learn from structural failures in concrete, steel and timber?* . s.l. : Lund University, 2007.
12. *Analysis of structural failures in timber structures: Typical causes for failure and failure modes*. Hansson, Eva Frühwald. 2011, Elsevier - Engineering Structures.
13. *Statistics of damages to timber structures in Germany*. Frese, Matthias and Blaß, Hans Joachim. 2011, Elsevier - Engineering Structures.
14. *Schäden an Holztragwerken*. Dröge, G and Dröge, T. Fraunhofer IRB Verlag, 2003, Schadenfreies Bauen, Vol. Vol. 28.
15. IPCC. *Climate Change 2023: Synthesis Report. Contribution of Working Groups I, II and III to the Sixth Assessment Report of the Intergovernmental Panel on Climate Change*. Geneva, Switzerland : IPCC, 2023.
16. One Click LCA Ltd. *Life Cycle Assessment for buildings - Why it matters and how to use it*. 2021.
17. Latteur, Pierre. *Cours de Structures en bois*. Louvain-la-Neuve : UCLouvain - EPL - GCE, 2023.
18. Dovetail Partners, INC. *Carbon in Wood Products - The Basics*. Minneapolis : <http://www.dovetailinc.org>, 2013.
19. *Advancing the circular economy and environmental sustainability with timber hybrid construction in South Korean public building*. Shind, Bigyeong and Kim, Sumin. 2024, Elsevier, Building and Environment.

20. Zeller, Vanessa, et al. *PROJECT REPORT 4V.1 - Life Cycle Assessment of Wood Buildings*. s.l. : Enecobois, 2015.
21. Nyssen-Dehaye, Damien. *Cours de Conception de l'architecture avec le bois*. Louvain-la-Neuve : UCLouvain - EPL, 2024.
22. FAO. *The State of the World's Forests 2024 - Forest-sector innovations towards a more sustainable future*. . Rome : s.n., 2024. <https://doi.org/10.4060/cd1211en>.
23. Goosse, Justin. *RoboTronc - Réunion mensuelle - Novembre - WP4 - Principe de l'ossature poutres/colonnes*. s.l. : UCLouvain, 2023.
24. PEFC. PEFC.org. [Online] 2024. <https://www.pefc.org/>.
25. Aicher, Simon, Reinhardt, H.-W. and Garrecht, Harald. *Materials and Joints in Timber Structures: Recent Developments of Technology*. s.l. : Springer, 2014. ISBN 978-94-007-7810-8.
26. Thyssen, Nelle, et al. *PROJECT REPORT 1V.2 - Wood Flow Analysis of the Walloon Region*. s.l. : Enecobois, 2014.
27. Duffield Timber. Hardwood vs. Softwood: What Are The Differences? [Online] 2024. <https://duffieldtimber.com/the-workbench/timber-trends/hardwood-vs-softwood-what-are-the-differences#:~:text=Hardwoods%20derive%20from%20angiosperm%20trees,is%20usually%20denser%20and%20stronger..>
28. *Water in wood: A Review of Current Understanding and Knowledge Gaps*. Thybring, Emil Engelund, et al. 2051, s.l. : Academic Editors: Paloma de Palacios and Francisco Garcia Fernández, 2022, Vol. 13. <https://doi.org/10.3390/f13122051>.
29. Goosse, Justin and Latteur, Pierre. *MISE AU POINT D'UN PROTOCOLE NON DESTRUCTIF DE CARACTÉRISATION MÉCANIQUE DE GRUMES*. s.l. : UCLouvain - EPL - GCE, 2023.
30. United States Department of Agriculture, et al. *Wood Handbook - Wood as an Engineering Material*. 2010.
31. Sanna, Fausto. *Designing timber buildings*. s.l. : The Crowood Press, 2022.
32. WoodWorks. *Mass timber - Design Manual - vol. 2*. s.l. : Think Wood, 2022.
33. Onate, Eugenio. *Structural Analysis with the Finite Element Method. Linear Statistics. Volume I. Basis and Solids*. . 2009. ISBN: 978-1-4020-8732-5 (HB).
34. *A state-of-the-art review of carpentry connections: From traditional designs to emerging trends in wood-wood structural joints*. van Nimwegen, Serena E. and Latteur, Pierre. UCLouvain, 2023, Elsevier - Journal of Building Engineering.
35. Mirianon, Florian, Fortino, Stefania and Toratti, Tomi. *A method to model wood by using ABAQUS finite element software*. s.l. : VTT Technical Research Centre of Finland, 2008.
36. Nilsen, Trygve Stenrud. *Numerical modelling of Wood Microstructure*. s.l. : NTNU - Trondheim, 2015.
37. *Numerical modelling of wood under combined loading of compression perpendicular to the grain and rolling shear*. Akter, Shaheda T., Serrano, Erik and Bader, Thomas K. 2021, Elsevier - Engineering Structures.

38. *Numerical modelling of timber and timber joints: computational aspects*. Sandhaas, Carmen, Sarnaghi, Ani Khaloian and van de Kuilen, Jan-Willem. Springer, 2020, Wood Science and Technology.
39. *Numerical analysis of civil engineering structures*. Almeida, Joao and Rattez, Hadrien. UCLouvain, Louvain-la-Neuve : s.n., 2023.
40. Dassault Systems Simulia. *Abaqus/CAE User's manual*.
41. —. *Abaqus Theory Guide*. 2016.
42. Wang, Mingqian and Song, Xiaobin. *Three-dimensional combined elastic-plastic and damage model for nonlinear analysis of wood*. s.l. : Journal of Structural Engineering, 2018.
43. Ma, Hua, et al. *Seismic Performance of Damaged Dovetail Joints with Different Damaged Degrees in Timber Frames*. s.l. : Hindawi, Advances in Civil Engineering, 2019.
44. Institut Belge de Normalisation. *Timber structures - Joints made with mechanical fasteners - General principles for the determination of strength and deformation characteristics*. Bruxelles : s.n., 1992. NBN-ISO 6891.
45. Jodidio, Philip. *100 Contemporary Wood Buildings*. Köln : Taschen GmbH, 2023.
46. Klanten, Robert and Stuhler, Elli. *Evergreen Architecture, Overgrown Buildings and Greener Living*. s.l. : gestalten, 2021.
47. *Argumentaire en faveur de l'utilisation du bois dans les bâtiments multiétagés*. Nyssen-Dehay, Damien. 2020, FormaWood.
48. *Fire Design of Mass Timber Members*. McLain, Richard and Breneman, Scott. 2023, WoodWorks - Wood products council.
49. *Demonstrating Fire-Resistance Ratings for Mass Timber Elements in Tall Wood Structures*. McLain, Richard. 2021, WoodWorks - Wood products council.
50. *Mass Timber Cost and Design Optimization Checklists*. WoodWorks. 2022, WoodWorks - Wood Products Council.
51. *Insurance for Mass Timber Construction: Assessing Risk and Providing Answers*. McLain, Richard and G. Brodahl, Susan. 2021, WoodWorks - Wood products council.
52. *How Can a Developer/Owner Get Started with*. WoodWorks. WoodWorks - Wood products Council.
53. A/S, ETA-Danmark. *European Technical Assessment, Screws and threaded rods for use in timber constructions*. 2019.
54. *Preliminary version of the new EN 1995: Eurocode 5 - Design of timber structures*. 2023.
55. *A holistic framework for designing for structural robustness in tall timber buildings*. Voulpiotis, Konstantinos, et al. 2020, Elsevier, Engineering structures.
56. *Construction Challenges and Solutions in Mass Timber: The Case of the Dengo Store*. Urbem. 2024, arch daily.
57. Kaufmann, Hermann, Hofmeister, Sandra and al., et. *Hermann Kaufmann Architekten - Architecture and Construction Details*. s.l. : DETAIL, 2023.
58. *Mechano-sorptive structural analysis of wood by the Abaqus finite program*. Santaoja, Kari, et al. 1991, ESPOO 1991, Vol. Research notes 1276.

59. HoutInfoBois. *Le plus haut immeuble de bureaux en bois à Bruxelles*. URL: <https://www.houtinfo Bois.be/realisations/le-plus-haut-immeuble-de-bureaux-en-bois-a-bruxelles/> : Last consulted: 25-02-2024, 2024.
60. Jaeger , Jean-Marc, et al. *QUAND LES IMMEUBLES DE GRANDE HAUTEUR SONT EN BOIS*. s.l. : Annales du Bâtiment et des travaux publics, 2017.
61. *Material model for wood*. van de Kuilen, Jan-Willem and Sandhaas, C. No. 2/3, 2013, Heron, Vol. 58.
62. *Engineered wood products and an introduction to timber structural systems*. Structural Timber Association. Structural timber - Engineering bulletin.
63. *Influence of anisotropy on the energy release rate $G(i)$ for highly orthotropic materials*. Kossakowski, Paweł Grzegorz. 2007, Journal of Theoretical and Applied Mechanics.
64. *Verification of orthotropic model of wood*. Obara, Paula. 3, 2018, Archives of Civil Engineering, Vol. LXIV, pp. 31-41.
65. Pierson, Alexis. *Utilisation d'indicateurs morphologiques pour définir des règles de dimensionnement de planchers composés de grumes*. Université catholique de Louvain : Ecole polytechnique de Louvain, 2021.
66. Kubba, Sam. *Handbook of green building design and construction : LEED, BREEAM, and Green Globes*. s.l. : Elsevier, 2012.
67. *Challenges and potential for human–robot collaboration in timber prefabrication*. Yang, Xiliu, et al. 2024, Elsevier, Automation in construction.
68. *The wood from the trees: The use of timber in construction*. Ramage, Michael H, et al. 2016, Elsevier, Renewable and Sustainable Energy Reviews.
69. *Environmental benefits of using CLT structure in midrise non-residential construction: an LCA based comparative case study in the U.S. Pacific Northwest*. Pierobon, Francesca, et al. 2019, Elsevier, Journal of Building Engineering.
70. WoodWorks. *Sustainability*. s.l. : <https://www.woodworks.org/why-wood/sustainability/>, Consulted in 2024.
71. International Timber. *How Can Timber Be Used To Reduce Environmental Impact Of A New Build?* s.l. : <https://internationaltimber.com/resources/how-can-timber-be-used-to-reduce-environmental-impact-of-a-new-build/#:~:text=Timber%20is%20made%20from%20carbon,adding%20to%20the%20greenhouse%20effect.&text=When%20used%20as%20a%20building,reduce%20emissions%20a,> 2020.
72. Timbeck. *Benefits of Using Timber*. s.l. : <https://timbeck.com.au/benefits-of-using-timber/>, 2023.
73. HoutInfoBois. *Etat de la construction bois en Belgique 2011-2022*. s.l. : HoutInfoBois, 2022.
74. —. *Les performances du bois*. s.l. : <https://www.houtinfo Bois.be/la-foret-et-le-bois/impact-environnemental-2/acv-epd/>, 2024.
75. Sathre, Roger and O'Connor, Jennifer. *A synthesis of Research on Wood Products & Greenhouse Gas Impacts, 2nd Edition*. Vancouver, B.C. : FPInnovations, Technical report TR-19R, 2010.

76. USDA - United States Department of Agriculture. *National Report on Sustainable Forests - 2010*. s.l. : Forest Service, 2011.
77. Charlier, Marion and Vassart, Olivier. *LGCIV2033 - Steel and composite structures*. UCLouvain : LCA and decarbonization, 2023.
78. EM-DAT: the OFDA/CRED International Disaster Database. *Number of extreme weather events globally, by year*. Brussels : UCLouvain, 2013.
79. Bravo, Felipe, et al. *Managing Forest Ecosystems: The challenge of Climate Change*. s.l. : Springer, 2008. ISBN 978-1-4020-8342-6.
80. *Forests, Wildfire & Rural Communities*. WoodWorks. 2024. <https://www.woodworks.org/why-wood/forests-wildfire-rural-communities/>.

ChatGPT has been used to rephrase some parts of the text but has not been used to calculate.

UNIVERSITÉ CATHOLIQUE DE LOUVAIN
École polytechnique de Louvain

Rue Archimède, 1 bte L6.11.01, 1348 Louvain-la-Neuve, Belgique | www.uclouvain.be/epl