FE ANALYSIS OF STEEL LINKS FOR SEISMIC RESISTANT TIMBER FRAMES

Author: Ingrid Krause de Almeida

Supervisor: Prof. Dr. Eng. Federico Massimo Mazzolani

SUSCOS Responsible: Prof. Dr. Eng. Raffaele Landolfo

University: UNINA – Università degli Studi di Napoli Federico II

University: University of Naples "Federico II"

Date: 03.02.2014

STATEMENT OF THESIS APPROVAL

*This thesis, prepared by Ingrid Krause de Almeida entitled '**FE Analysis of Steel Links for Seismic Resistant Timber Frames**', is approved in partial fulfillment of the requirements for the degree of Master of Science by the following faculty members served as the supervisory committee chair:*

______________________________ Chair ________________
date

______________________________ Member ________________
date

______________________________ Member ________________
date

student's signature ______________________________

date of submission ______________________________

ABSTRACT

This thesis is part of a research on special steel connections for seismic resistant timber structures which aims to present a reliable design concept to increase the use of timber in multi-storey buildings. By clarifying the response of this kind of structures, the obtained detailed results should be useful to set-up appropriate design rules, since Eurocode 8 part 8 does not give detailed provisions in these cases.

The proposed technology consists of a steel profile inserted in a beam-to-column joint of a timber frame. This element is designed to have enough ductility and capacity for dissipating the seismic energy of a MRF, acting as the so-called "seismic link". This element, also placed in column-base joints, should ensure the integrity of the global structure and enhance the system into a high q behavior factor comparable to the values used for steel frames.

The focus of this thesis is the development of a finite element model in Abaqus software to predict the response of the seismic link connected to a glulam beam with edgewise plates and self-tapered inclined screws. The verification of the required overstrength of the connected elements to the steel link was validated by modelling the available experimental results and can sustain the analytical analysis made for the proposed joint. All the issues related to the development of these models, specially the simplifications made for modelling such complex material as wood, are here described.

Experimental campaign of the presented model is under development at the University of Trento and its results will be used in a further stage of this research for numerical calibration.

CONTENTS

1. INTRODUCTION

1.1. HISTORICAL BACKGROUND

The **earliest seismic design** for buildings in Europe was developed after the earthquake of Lisbon in 1755. Based on the observation of the still-standing buildings, a structural system was established for the reconstruction of the city. This technology consisted of a 3D timber bracing system filled with masonry called "gaiola pombalina" (Branco, 2009). These walls can be found all over Portugal and recent experimental tests confirmed their efficiency during seismic events since they can still keep their own integrity after many cycles of loading. The result of one tested frame can be seen on the top of figure 1 (Gülkan & Langenbach, 2004).

figure 1 - Gaiola Pombalina wal (top) and Hımış wal (bottom) (Gülkan & Langenbach, 2004)

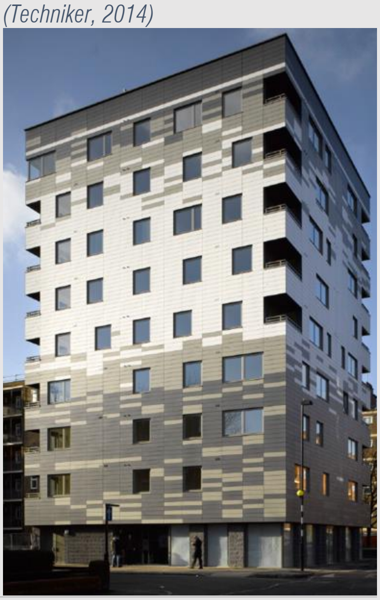
Following the experiences in Portugal, Italy also conceived the same system named as "Casa Baraccata", which was imposed by the Bourbon's government, just after the earthquake of 1783 in Calabria. Buildings constructed with the referred system were assessed by Ruggieri (Ruggieri, Tampone, & Zinno, 2013) in the last year. He presents the masonry as responsible of a limited displacement of the timber frame, which can continue to behave elastically during an earthquake. This system based on half-timbered structures was described as a construction specification in the **first rules for seismic construction** for hazard mitigation created in Italy on the nineteenth century.

The Ottoman-style houses, the hımış, present almost the same construction concept and have also survived several earthquakes until now. The system consists of a flexible-frame-with-masonry-infill (figure 1 - bottom) placed over ground floor walls made of heavy stone. The top storey, larger in plan than the bottom one, is supported by cantilever joists that create compression forces on the unreinforced bottom wall assigning to it lateral resistance (Gülkan & Langenbach, 2004). Their structural system behaves in a plastic way showing a ductile response, a fundamental characteristic to seismic resistant buildings.

A century later, the use of reinforced concrete and steel as structural materials was spread all over the world and dominated the construction fields because of market forces and cultural reasons (Gülkan & Langenbach, 2004). However, during the past years, timber has being raising in importance again and its performance facing seismic events are under research.

1.2. MOTIVATION OF THE RESEARCH

Residential Building – London, 2008 (Techniker, 2014)



Residential Building – Melbourne, 2013 (Thomson, 2014)

figure 2 - Tall timber buildings

In a short period of time, **tall timber structures** started to appear in different parts of the globe and the addition of one storey is a landmark in the Engineering history. In 2008, a timber building nine-storey high was constructed in London in a Podium structure, which means for this case, a cross-laminated timber (CLT) panel structure over a reinforced concrete ground floor (Techniker, 2014). From the same wood supplier: KLH, the world's tallest timber building with ten storeys was complete in Melbourne on December 2012 also in podium system (Fountain, 2013). Both of the buildings can be seen in figure 2, it should be remarked that they present no visual distinction from the finished result with another structural system material.

These buildings are result of many researches dedicated to the progressive knowledge of the seismic behavior of multi-storey buildings in **massive timber structures made from CLT panels**. The SOFIE is one of them. It is an Italian project that investigates these kind of structures and in 2007, performed a full scale test on the 3D shaking table of the National Institute for Earth Science and Disaster Prevention (NIED) in Tsukuba, Japan (figure 3). The tested building of seven storeys and total height of 23.5 m didn't show permanent deformations and maintained its shape even after series of loadings based on high destructive earthquakes. The results indicate a behavior factor of 3 and a good response of the system when subjected to accidental lateral forces (Sandhaas & Ceccotti, 2012).

The material wood itself is ductile in compression but quasi-brittle in tension and shear, that is why under cyclic loads the plasticity is given by steel connections. Because of the low number of connections, heavy timber frames have a different when behavior facing earthquake events, but up to the present **few researches** are dedicated to its assessment.

In order to improve the seismic behavior of timber frames, steel elements can be inserted in the system to act as the ductile element protecting the brittle material structure. One solution for **seismic resistant timber frames** has its inspiration in the "dog bone" for steel structures. The concept of localized damage is maintained when a weak element is inserted in the structure. Therefore steel profiles with lower strength than the other elements can be placed in the timber frames in particular locations: beam-to-column joints and column bases, forcing

figure 3 – Shaking table test – SOFIE project (Magrone, 2012)

the structure to fail in a global plastic mechanism. The plastic hinges are designed to occur in the steel links and the timber elements are kept in the elastic range.

A research like this was carried out by the **University of Trento** (Tomasi, Zandonini, Piazza, & Andreolli, 2008) in 2007. They developed a steel profile connected by end-plates and glued longitudinal steel bars into timber elements. The test results showed yielding of the bars and bending of the endplate but no deformations, occurred in the link because of the test setups had stiffeners in the endplate (Andreolli, Piazza, Tomasi, & Zandonini, 2011). Their focus was on characterizing of the connection steel-to-timber. In the same concept, a new connection was designed by the **University of Naples** "Federico II" consisting of a steel link connected by edgewise plates and inclined screws to a timber beam.

The present thesis is focused on this subject: the characterization of the steel link behavior in a beam-to-column joint by numerical analysis in Abaqus software.

1.3. WHY TO USE TIMBER

In times of climate changes, **environmentally responsible** building materials are under the spotlight. Considering a good forest management, when compared with concrete and steel, wooden materials pollute less the water resources and have a lighter carbon footprint. To give an example, a research on a 42-storey building made from Concrete Jointed Timber Frame in Chicago was produced by SOM and described in a report published on 2012 (Skidmore, 2013). The result of the study is a reduction of 60 to 75% of the carbon footprint if the building was made from steel or reinforced concrete.

When a Life Cycle Assessment is performed, timber structures can be classified in a cradle to cradle system because wood can be reused, recycled and it is biodegradable (CWC - Canadian Wood Council, 2004), which means **no waste** is produced.

Beyond that, timber structural elements have a very high strength to weight ratio, what **reduces costs** of transportation and also of foundations. Derived from this advantage, there is a high level of prefabrication of the timber systems, reflecting in the quality and reducing the erection time and the waste of the construction keeping the cost competitive in the marketplace (Rosenfield, 2012).

In addition to that its smooth surface can be visible from the inside, eliminating the necessity of finishing materials like plasterboards or leveling mortars. This is one of the main characteristics that makes timber structures popular choice among **architects**. But their interest is also on the flexibility of shapes and and in the aesthetic color and pattern given by the natural properties of wood. Renzo Piano, Pritzker-Prize Laureate architect, is recurrently choosing timber structures for his works (figure 4). Just to point some of his projects in Italy, the Auditorium Parco della Musica of Rome, open in 2002, has its roof supported by timber curved beams. Called by the same name, the auditorium for L'Aquila city finished in 2012 was made of entire 18 m wooden façades designed to be seismic resistant (F&M-ingegneria). In 2013, Le Albere, a residential 5-storey building from timber frames, was concluded in Trento.

Rome - Auditorium *L'Aquila – Auditorium* *Trento – Residence*

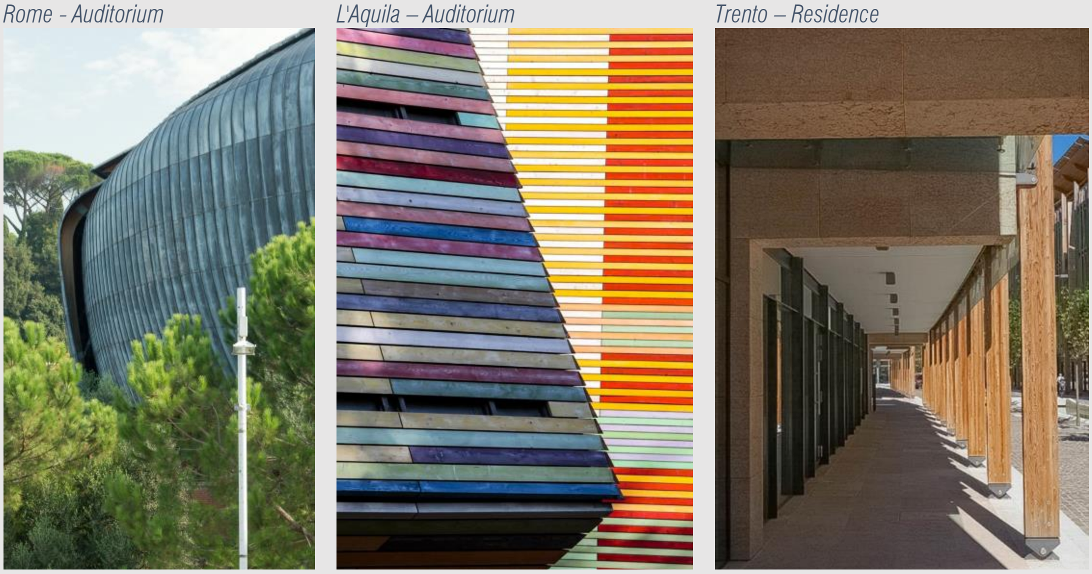


figure 4 - Renzo Piano's timber work - author's photography, (F&M-ingegneria) and Ioan Andreescu's photography

Considering **energy performance** during the operation, wooden structures can also result in a lower demand on cooling and heating systems because it has low thermal conductivity.

One major concern lies on the **durability** of wood because of termite attack and biological degradation. But these problems can be mitigated by a careful design and regular maintenance. The historical buildings described in the first section of this Introduction can show that, with moisture control, timber structures can survive centuries.

One of the reasons for limiting the number of storeys in design codes is the **fire resistance**, because wood is a combustible material (Frangi, Fire Resistance of Timber Structures, 2009). However in case of heavy timber frames, the structural members exposed to fire develop a char layer that protects the residual inner cross-section from the elevated temperatures. That is the reason why wood is considered as a self-protecting material. If the structural material is kept safe, the attention should be concentrated on keeping the steel connections safe. Therefore, connections with slotted-in plates have better fire performance than side steel plates. Hence, fire safety, as well as many other issues, is more dependent on the concept of the design than on the structural material (Frangi, Fire resistance assessment of timber structures, 2012).

Because of the advantages described above, governments like in France are supportive for the increasing usage of timber structures for all the related agents: production, manufacturing and construction (République Française, 2010). They search to eliminate the existing obstacles for the **development of its industry**, to improve training, to inform and aware the involved public, to provide financial support and to promote visibility for the projects that are using wooden structures (CNDB, 2011-2012). Hence there is an increasing tendency for new wooden buildings all over the world.

1.4. TIMBER CONNECTIONS

As any other structural system, there are particular conditions inherent to the design process of timber structures. First of all the design concept should be based on the fact that **wood** is a heterogeneous and anisotropic material. Its properties vary in function of the moisture content and the duration of the load. And more, it presents a ductile behavior when subjected to compression loads and a quasi-brittle failure under tension loads.

There are some solutions that have been developed to improve the structural properties of timber members, like Glulams, CLT and solid beams with FRPs, not to mention the use of hardwoods. In spite of all these new technologies, timber joints are still the **critical part of the structure**, they characterize 80% of their failure. Their mechanical performance is responsible for the continuity of the strength in a structure and in most part of the cases they require oversized connected elements (Santos, Jesus, Morais, & Lousada, 2008).

The **load carrying capacity** of timber connections is dependent on the orientation of the load, on the properties of the connected timber elements, on the fasteners properties and on the type of interaction. It is a system performance that should be characterized and well predicted in order to result in a reliable design.

The development of **numerical analyses** are one possible way to predict the behavior of these elements. However the connection should be discretized and its components validated so that their combined behavior can be assessed and as well calibrated. It is a process dependent on execution of tests, but the fine calibration is difficult to achieve, especially in case of wooden materials. Nevertheless, the importance of obtaining a valid numerical model for connections is the possibility to optimize and simplify the solutions.

Speaking about seismic loads, connections also play an important role in the structure response. They are the main responsible elements for providing the required ductility for the whole structure and prevent premature collapse keeping the integrity of the building. This lead the research to the next section: Earthquake Resistance.

1.5. EARTHQUAKE RESISTANCE AND ITS CODIFICATION

The European earthquake standard 8 – part 1 gives general rules for designing seismic resistant buildings in a **performance-based approach**. Considering social and economic questions, its concept relies on the probability of the event to happen and the importance of the building's use and the expected severity (SZS, 2011). The structural design should reduce the risks of an earthquake ensuring the protection of human lives, the limitation of the damage and also ensure that structures important for civil protection remain in operation.

First of all, the **seismic actions** should be defined based on the natural frequency of the structure, the ground conditions, topography and the seismic zone, which is dependent on the local hazard zonation maps related to a referential return period of occurrence. Therewith the hazard is quantified in one single parameter: the reference value of the peak ground acceleration (PGA).

It is also included in the code two types of **safety verifications**: the ultimate limit state (ULS - no-collapse) and the damage limitation state (DLS). In a general description, for the former state, the resistance of the structural elements should be lower than the calculated actions and for last one, the structure should present interstorey drift values below the limits.

For the **design of structures**, EN8 defines the basic concepts which are mainly the adequate exploitation of ductility and the definition of hierarchy resistance between the component elements of the structure. The ductility in general words is the property of a structure to deform without brittle failure beyond its elastic limit (Tomasi, Zandonini, Piazza, & Andreolli, 2008). The second requirement is intrinsically dependent on the former, it means that the elements responsible for the ductility have to exhibit a lower resistance in comparison with its connected elements so that locally and globally the structure can develop the desired mechanism.

The code dedicates one section for defining **specific rules for timber structures**, but considering that its last version was published on 2004, it doesn't cover many new technologies developed up to the moment. It is worth pointing that there are no design details or parameters defined specifically for CLT buildings (Sandhaas & Ceccotti, 2012). Another question involving lack of data in the code concerns hyperestatic portal frames with doweled and bolted joints. The standard limits in 2.5 or 4 the value of the behavior factor for this typology, according to its ductility classification divided in medium or high capacity to dissipate energy. The difference can just be set by test results on single joints, part or whole structures focusing on the rotation ductility capacity of connections.

In order to insert more details about a **new typology in EN8**, the proposed technology should be characterized and better quantified by the development of experimental campaigns followed by calibrated numerical models. These computational step is extremely useful for changing variables such as building geometries and earthquake loads offering an optimization tool and enough data so that the solution can be classified as reliable (Sandhaas & Ceccotti, 2012). With this procedure it is possible to establish new seismic design parameters for codification of the type of structure.

This thesis is situated in this context, the elaboration of the FE model to improve the quantification of the seismic behavior of the **seismic steel links** connected to timber beams in a moment resistant frame (MRF). The results in this early stage of the research show that this new technology, if carefully designed, can enhance values of behavior factors for heavy timber frames to one comparable to steel frames.

2. THE PROPOSED SEISMIC DEVICE

2.1. GLOBAL ANALYSIS

GENERAL INFORMATIONS

This work is part of a research undertaken by the University of Naples "Federico II" on the characterization of seismic joints for timber structures. The work was carried out in cooperation with the Eng. **Roberto Tartaglia**, who developed the global seismic analysis and the design of the studied connection as part of his Master Thesis, which will be published in the near future.

DESIGN CONCEPT

As case study a 2D seismic response of a 5 storey-2 span **moment resisting frame** (MRF) for residential purposes has been considered. Beams cross-section are 500 x 240 mm^2 and columns cross-sections 300 x 240 mm^2. The spans have 6 m each. The ground-floor is 4 m high and the storeys above are 3.5 m high. The global arrangement can be seen in figure 5.

The typology of the structure offers two main advantages. First the architectural benefit of having a span with no obstacles like bracings providing architects more freedom in façade and in layout designs. Second, the possibility of enhancing the **ductile response** of the structure, which is made from a brittle material, thanks to a steel seismic link.

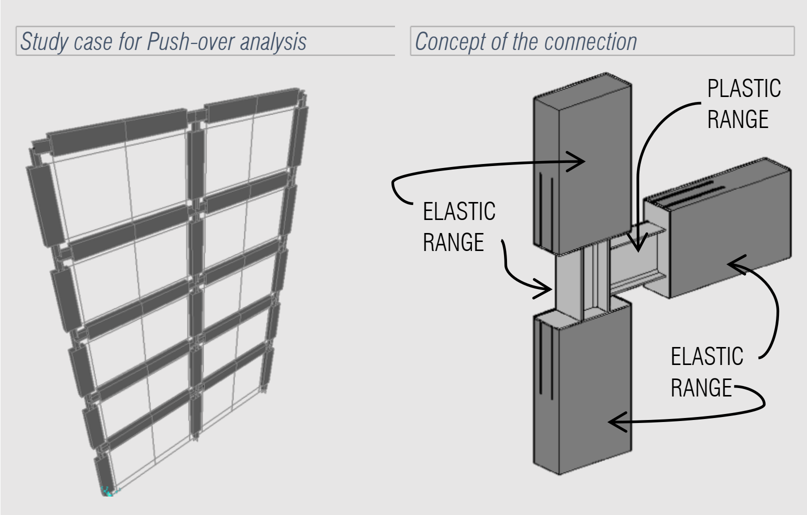


figure 5 - Global arrangement and concept of connection

Speaking about load redistribution, EN8 emphasizes that the design should pursue the **structural simplicity**, in which there are clear and direct paths for the transmission of the seismic forces in order to reduce uncertainties and increase the reliability of the predicted seismic behavior. Situated in this idea, MRF offer very clear configuration and distribution of stresses.

The designed timber members are made in **homogeneous glued laminated timber** with characteristic bending strength of 28 MPa. The use of glulam have the intention to minimize the influence of local defects intrinsic of wooden materials (Sousa, Branco, & Lourenço, 2013). The choice for the homogenous product allows a more continuous distribution of stresses in the cross-section because each lamella has the same material properties. Considering seismic applications, homogeneity of the strength is essential condition to provide symmetry to the structural element.

The **mechanical behavior** of wood varies depending on the load: ductile under compressive loading and quasi-brittle under tension or shear loading. Focused in the improvement of its bending and axial stiffness, for example some technologies have been developed using Fiber-Reinforced Plastics (FRPs). When the energy dissipation of a timber system is dependent on its compression ductile behavior, during a seismic event, there will be irreversible crushing of the fibers and the degradation of the element stiffness (Reynolds, Harris, & Chang, 2012). Moreover, when some of the fibers are damaged in compression and there is an inversion of direction of the load, these same fibers have really small resistance in tension (Piazza, Tomasi, & Modena, 2005). Therefore, the ductile response of timber under compression loads cannot be explored for cyclic loading.

In another dissipation model, light timber frame structures dissipate earthquake energy by yielding their numerous steel fasteners (Andreolli, Piazza, Tomasi, & Zandonini, 2011). After a seismic event, **damage** can be observed in the connections and in the embedment zone of timber elements (Sandhaas & Ceccotti, 2012). In terms of rehabilitation, steel can be easily repaired, but the damage in embedment of timber are unrepairable.

In a way to solve these issues and still provide a seismic resistant timber structural system, the proposed technology explores the ductility of steel. The **steel links** placed in the frame can localize the plastic deformations and offer the possibility of easy repair after damage. The proposed technology can satisfy the code's requirement in the concept of a capacity design ensuring that the weakest elements in the structure have the required ductility for the whole system. In order to allow the formation of the plastic hinges in the links, the other elements should present enough overstrength to prevent premature brittle failure modes.

For the studied frame the seismic links are made of IPE 330 profiles for beam-to-column joints and HEB 240 for base-column joints. Their position induces the pattern of the plastic hinges and of the desired global mechanism.

PUSH OVER ANALYSIS

The pushover analysis is a **non-linear static analysis** predicted by EN8 performed as monotonically increasing horizontal loads on structures under constant gravity loads. As it was mentioned before, the global analysis of the proposed typology was carried out in a previous stage of the research. It was performed in SAP 2000 software (version 14.2) in order to estimate the expected plastic mechanism and the evolution of the damage distribution. It also aimed to estimate the behavior factor q in a way to compare the response of the system to the upper limits established by the European standards. This value is representative of the ductility and energy dissipation of the building. It gives the possibility of assessing the nonlinear behavior of a structure by developing a simplified linear verification where the applied seismic forces are reduced by the q value (Sandhaas & Ceccotti, 2012).

Following EN8 procedure, the structure was analyzed in a planar model with fundamental period of vibration of 0.38 s for the considered direction, which value was taken from the first vibration mode calculated through a modal linear analysis. The peak ground acceleration used was 0.35g, as if the building was placed in the highest seismic zone of Italy. The reference return period of the seismic action was taken as 50 years. These values were used to obtain the seismic **base shear force**, load that is distributed in horizontal forces over the storeys for the development of the pushover analysis. The results of the analysis are represented in capacity curves: the control displacement as function of the base shear force increase.

For the development of a performance based design, the American Standard FEMA 356 was considered and values for the Damage Limit State (DLS) were defined in pushover curve for the Multi-Degree-of-Freedom (MDOF) model. These bounds multiplied the result into four different levels of damage and each result in terms of MDOF curve was transformed in an equivalent one defining a Single-Degree-of-Freedom (SDOF) system. After this, the response can then be simplified in a **bilinear force-deformation relationship**, using the secant stiffness to the yield point as the elastic stiffness of the curve to characterize the post-yield behavior of the seismic link. This methodology can be followed graphically in figure 6, where the method for defining the MDOF with FEMA 356 limitations is described for a study case on 2-storey 1-span building.

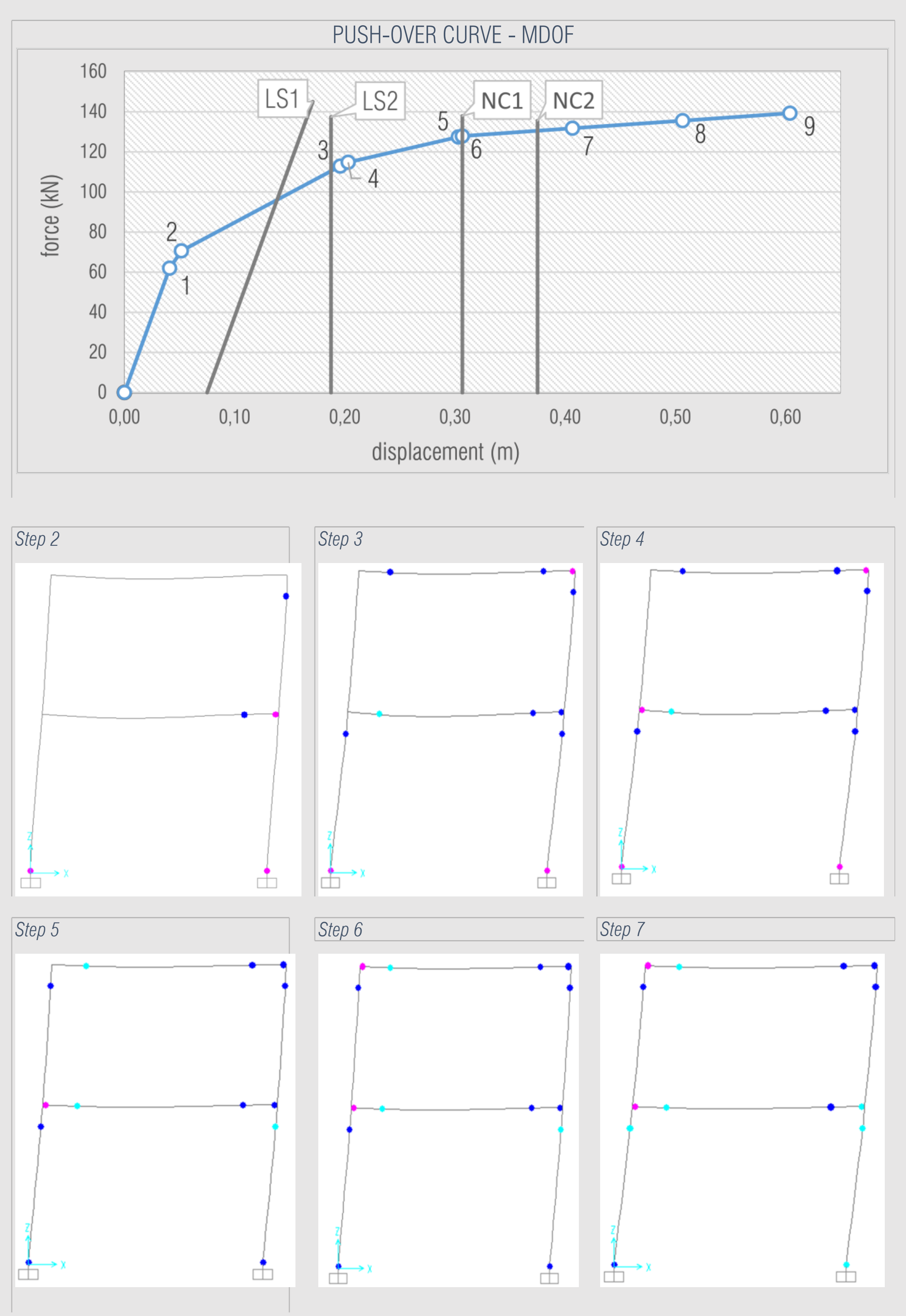


figure 6 - Push over curve and location of plastic hinges for the study case of 2-storey 1-span building ((Mazzolani, Faggiano, & Tartaglia, 2013)

The lowest value of the **q factor** acquired from the analysis interpretation of the bilinear curves for the SDOF in the performance based design was 6, giving a good indication that the system's response to lateral loads is comparable to steel MRFs and that the structural type would be underestimated by EN8. It can be concluded in this first phase of the studies that the new technology is possible to be built in earthquake-prone areas and that the research can proceed to define results with different geometries and building masses in order to have a final number of behavior factor (Sandhaas & Ceccotti, 2012)

2.2. DESIGN OF CONNECTIONS

PREVIOUS RESEARCHES

CONNECTION WITH GLUED-IN BARS

The first step considering the insertion of seismic steel links in heavy timber frames was performed by the University of Trento, research that was already mentioned in the introduction of this thesis. The developed technology consists of a seismic link made of steel profile welded to an end-plate **connected by steel bars glued** inside a glulam beam parallel to grain (see figure 7) (Andreolli, Piazza, Tomasi, & Zandonini, 2011). The experimental campaign was preceded by an analytical design based on the component method described in EN 3-1-8, the European Standard for the design of steel joints.

For the test set-up, a **cantilevered configuration** was created in order to produce bending and shear in the joint. Therefore the steel stub was rigidly fixed and forces were applied on the other end on the timber member. The specimens were made with different end-plate thickness with values of 6, 8 and 10 mm. As it was expected that shear forces would generate plastic deformations in the embedment zone of the timber, other specimens were made with steel plates glued inside slots grooved in the timber elements. For this arrangement, specimens were manufactured with end-plate thickness of 6, 8, 10, 15 and 20 mm.

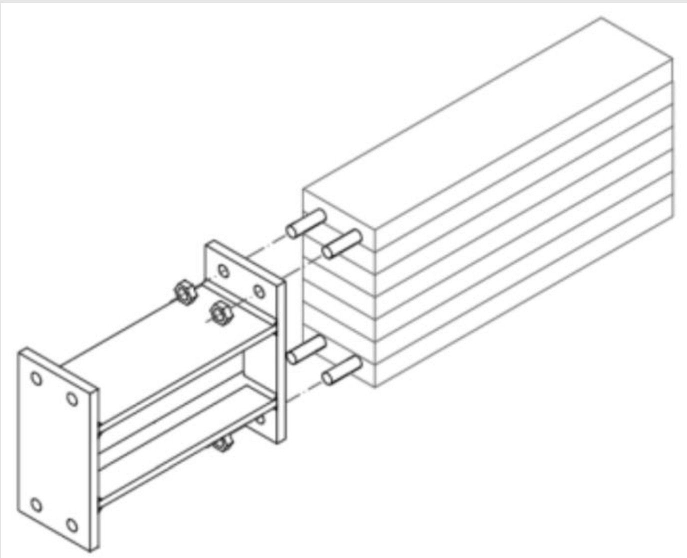


figure 7 - University of Trento's tested seismic link (Andreolli, Piazza, Tomasi, & Zandonini, 2011)

Despite of this differentiation the tests showed that the specimens without the glued-in plates could transfer the **shear loads** through the bars developing the same failure modes as the correspondent specimen with the plate, reaching equivalent values of moment resistance and rotation capacity.

The same configuration of specimens was tested under **monotonic and cyclic loading**. The monotonic experimental results presented a good correlation with the analytical values calculated for strength and rotation capacity and developed the same failure mode as predicted (values described in table 1).

The expected brittle failure for the glued-in bars placed parallel to-grain in timber members is related to the splitting of wood due to tension perpendicular to-grain. It can also develop failure in the interaction of the glue and connected elements creating a tensile failure where it can be observed splitting and bar slip. But none of these problems associated with brittle behavior could be seen in specimens subjected to monotonic loads. No collapse was related to the connection bar-timber, the collapse modes were related to the **yielding of the end-plate, bar yielding** in presence of prying forces and bar failure for the specimen with end-plate 20 mm thick. These last mentioned specimen could not develop the required plastic deformation capacity for an adequate seismic design.

Specimen	End-plate steel measured mechanical properties			Failure mode of T-stub in tension		Moment resistance: kNm		Rotation capacity: rad	
	ε_u: %	f_y: MPa	f_u: MPa	Theoretical	Experimental	Theoretical	Experimental*	Theoretical	Experimental†
P6-sp	38·5	299·6	377·2	1	1	7·08	15·07	0·15	0·20
P6				1	1	6·56	8·85	0·15	0·15
P8-sp	38·6	275·5	367·5	1	2	10·66	16·59	0·16	0·19
P8				1	2	10·51	14·99	0·16	0·21
P10-sp	45·9	256·1	374·0	2	2	15·37	17·14	0·14	0·12
P10				2	2	15·80	19·85	0·13	0·18
P15-sp	35·7	295·5	383·1	2	2	21·82	24·54	0·13	0·14
P20-sp	36·4	278·2	400·6	3	3	28·51	32·29	0·06	0·07

* Maximum value
† Displacement at failure

table 1 - Comparison of theoretical and experimental results of the joint (Andreolli, Piazza, Tomasi, & Zandonini, 2011) with indication of the results used for numerical calibration of Abaqus model

This research opened possibilities for the development of such ductile joints fulfilling the concept of protection capacity, keeping the timber member and connections in elastic field preventing premature collapse of the structure due to brittle failure.

SCREWED CONNECTION WITH SLOTTED IN PLATES

Following the same concept of energy dissipation, another connection for single-sided beam-to-column was developed by University of Naples as a preliminary design concept for improving the proposed connection. The structure in which this joint is addressed is the same as the one described in the previous section of this thesis, the 5 storey-2 span building. Aiming to make an introduction for the chosen solution and provide **comparative values** between the different connections, the analytical design will be described briefly in this section.

A column stub welded to a horizontal steel profile was designed to act as the ductile element of the joint. The full strength connection consisted of an end-plate welded to a couple of perpendicular **plates inserted** in the timber beam. Self-tapered screws were designed to connect the different materials and to be drilled in situ.

The global analysis was performed with beam links made by steel profile IPE 330 and steel grade of 235 MPa. Its resistance was used as reference value of to create an overstrengh between the link and the connected elements. The designed action used for the screwed connection was based on the plastic **moment** of the link multiplied by the amplifying parameters described in equation 1 given by EN 8 in order to provide local hierarchy and ensure the localized plastification in the link.

$$M_{Ed} = 1.1\,\gamma_{ov} M_{pl.IPE330}$$ *equation 1*

Defining in numbers the equation 1, the bending moment applied on the connection was based on an increased value of the resistant plastic moment of the link in 37.5 %.

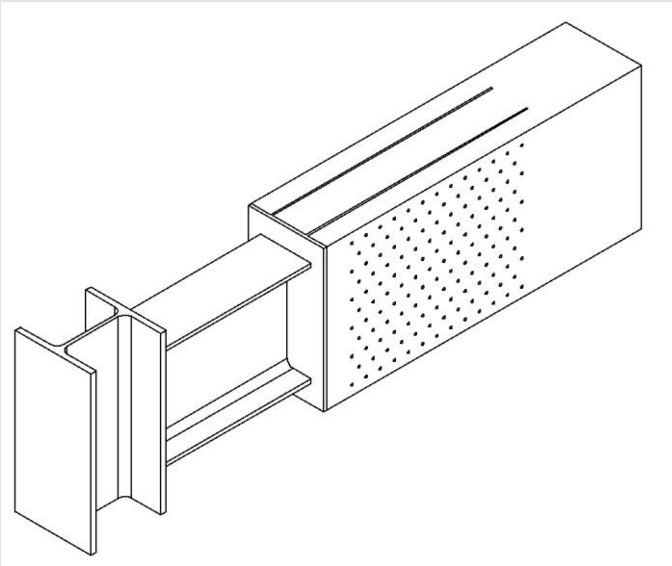


figure 8 – Screwed connection with slotted-in plates

The value for the applied **shear force** was taken as 100 kN, an output data from the pushover analysis. The **axial force** was considered as transferred integrally by the timber contact with the end-plate and was not included as action for the connection design.

The Rotho-fixing self-tapping screw CS 100255 with 7 mm of diameter, which is the maximum available, and 233 mm of length was chosen for this connection once it can be self-drilled into the two steel plates with 5 mm of thickness. These fasteners are loaded perpendicularly to theirs axis and are divided in four shear planes. The **load bearing capacity** and failure mode of the connection is obtained in function of the yield moment of the fastener and the embedment strength of the timber, parameter that depends on the angle of the applied force and the density of the wooden material, in this case GL28h. This analytical procedure followed the specific European Standard for the design of timber structures: EN 5.

For a moment resisting connection, a rectangular staggered pattern for the screw distribution induces the **greatest combined applied force** to happen in the fasteners more distant from the centroid of the group of screws. The resistance of this fastener was verified and the failure mode for double shear connections defined by developing one plastic hinge in the screw. The used screw type is threaded in just a small part of the shank, therefore their withdrawal capacity contribution for the rope effect was unconsidered in the calculations.

The final number of screws necessary for this connection was **138** respecting the minimum spacing established by the code to avoid premature splitting. The length of the internal plates was defined as 700 mm and the width in the same dimension as the height of the timber beam. The characteristic resistance of the connection was transformed in design value through the equation 2 based on the EN 5 requirements.

$$F_{V.Rd} = \frac{k_{mod} F_{V.Rk}}{\gamma_M}$$ *equation 2*

It is important to mention the origin for the **modification factors** placed in the equation 2. The one represented by the symbol k_{mod} is representative of the load-duration and moisture content, this value according to EN 5 is 1.1 for in cases involving instantaneous actions, adequate to seismic design. The partial factor for wood properties accounting uncertainties and dimensional variations, should be taken as 1.3 for connection design according to EN 5, however EN 8 allows the reduction of the material factor to 1.0, if the structure is inserted in a dissipative structural concept for ULS verifications. Based on these numbers, the resistance of the connection can be considered as 10% higher than its characteristic value.

PROPOSED CONNECTION

The proposed connection aims to **optimize** the solution described in the previous section, by exploring the use of inclined fully threaded screws, which produces an improvement of resistance and of stiffness (Crosatti, Piazza, & Tomasi, 2009).

A research on timber-to-timber connections with inclined screws was performed by Bejtka and Blass in 2002. They observed that the fastener's load carrying capacity reaches its maximum when the angle to the grain is 30º, providing an increase in 50% of the value of a screw loaded perpendicular to its axis. The **ultimate withdrawal capacity** was proven to be reached, independent on the screw diameter, when the fasteners are placed with inclinations higher than 30º. This conclusions were just considered in the EN 5 in 2008, when the procedure for assessing the withdrawal capacity of axially loaded screws was altered.

When the screws are placed at an inclination, its axial force develops a reaction force component perpendicular to the contact surface between the connected elements, generating **friction** forces in the opposite direction of the loading. This aspect can be pointed out as one more advantage for the use of these kind of solutions.

After the description of such advantages, the first connection system was considered uneconomical and another solution was designed using full threaded screws inclined at 45º. The same system can be found in the nodes of the ENEL's **Coal Storage Dome** for the Power Plant Brindisi Sud, under construction in the south of Italy, designed by H.E. Lüning Consulting Civil and Structural Engineers from the Netherlands (figure 9 and figure 26).

figure 9 – Edgewise connection of ENEL's Dome - University of Trento (Mazzolani's photography)

The proposed technology has the same concept of the previous studies already mentioned. The steel links are responsible to **dissipate the seismic energy** of the structure. Considering that the local plastic redistribution of stresses determines the resistance of the joint, the material assigned to the link should have sufficient ductility, this is one of the reasons why the steel grade of 235 MPa was selected for this element (Jaspart, Demonceau, Renkin, & Guillaume, 2009). In respect to local hierarchy, the other elements should be designed with a minimum standard required **overstrength** in order to enable the development of the plastic hinges to happen in the link and finally the achievement of a global failure mechanism.

For these reasons, the connection of the link to the timber member should not trigger any plastic deformations and should to be designed as a **rigid, full strength connection** with high bending resistance and rotation stiffness, ensuring that the timber will remain in the elastic range of analysis and assure that after the event, any embedment damage will occur.

The final design of the joint consists of a steel profile welded to an end-plate also welded to an **edgewise connection** with inclined screws. Moreover, the plates in S235 were designed with 20 mm of thickness and the screw head has full bearing area onto the predrilled connected plate. The selected screw was taken from Rothoblaas' catalog, named as VGS11300, which means 11 mm of nominal diameter and 300 mm long. Its geometrical characteristics are flared head, full length thread and self-drilling properties.

The design procedure of EN 5 considers this kind of connection in a combined behavior as is the screws were loaded laterally and axially. The systems' load carrying capacities, dependent on the timber embedment, screw properties and angle to-grain of the fastener's and of the load, should be considered separately and the final verification should fit inside the **interaction limit** defined by the expression described below.

$$\left(\frac{F_{ax,Ed}}{F_{ax,Rd}}\right)^2 + \left(\frac{F_{v,Ed}}{F_{v,Rd}}\right)^2 \leq 1$$ *equation 3*

The first element of the equation 2 is composed by the ratio between the design axial force on fastener ($F_{ax,Ed}$) and the design value of axial withdrawal capacity of the fastener ($F_{ax,Rd}$). The second member by the ratio between the design shear force per shear plane of fastener ($F_{v,Ed}$) and the design load-carrying capacity per shear plane per fastener ($F_{v,Rd}$).

The design moment is the same as described in the previous section respecting the overstrength provided by equation 1. The same result value was taken because the steel profile described for the link remains the same: IPE 330. The **tension force** acting in the shear plane created by the bending moment, action that will be carried out by the screwed connection, while the compression is assumed to be transferred by contact of timber directly to the end-plate.

The standard load carrying capacity in case of laterally loaded screws depend on the embedment strength of timber, the yield moment of the fastener and mobilizes 25% of the axial withdrawal capacity ($F_{ax,Rk}$) if the failure mode is related to the yielding of the fastener. The governing failure mode for this connection is predicted as the development of 2 plastic hinges in the screw. EN 5's expression for the definition of its characteristic value per fastener is given by equation 4 for steel-to-timber connections in single shear and thick plates.

$$F_{v,Rk} = 2.3\sqrt{M_{y,Rk} f_{h,k} d_{ef}} + \frac{F_{ax,Rk}}{4}$$ *equation 4*

Being $M_{y,Rk}$ the characteristic yield moment of fastener, $f_{h,k}$ the characteristic embedment strength, and d_{ef} the effective diameter of the screw, described in section 8.7.1 (3) as being 1.1 times the thread root diameter.

The withdrawal resistance of the fastener is the only parameter influencing the axial load carrying of this kind of connections. Its expression, described in equation 5, was defined the revision of the code and depends on the angle α between the screw axis and the grain direction, considering the established a minimum value as 30º.

$$F_{ax,k,Rk} = \frac{n_{ef} 0.52\, d^{0.5} {l_{ef}}^{0.9} {\rho_k}^{0.8}}{1.2(\cos\alpha)^2 + (\sin\alpha)^2} \qquad \text{equation 5}$$

The equation 5 is dependent on the outer thread diameter of the screw (d), the effective length of the screw inside the member (l_{ef}), the characteristic density of the timber element and the angle between the force and the direction of grain (α).

The Crosatti et all., 2009 - Since the minimum screw spacing in the plane parallel to the grain was respected, the theoretical effective number of fasteners (n_{ef}) calculated according to EN 5 was equal to the actual number of fasteners n (Crosatti, Piazza, & Tomasi, 2009).

Moreover the expression given in equation 2 was used to transform the characteristic values of resistance of the connection into a design one and the equation 3 was verified.

The final design configuration resulted in 7x4 screws per plate being 56 the total number of required screws. This result is less than half of number of fasteners necessary for the previous connection configuration, therefore the installation costs are directly reduced. The drawings presented in figure 10 and figure 11 characterize the set up

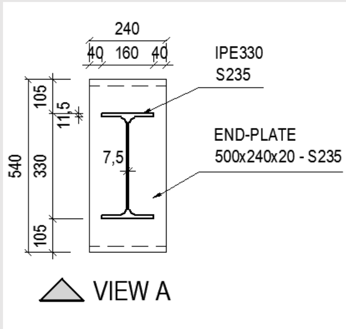


figure 10 – Edgewise Connection

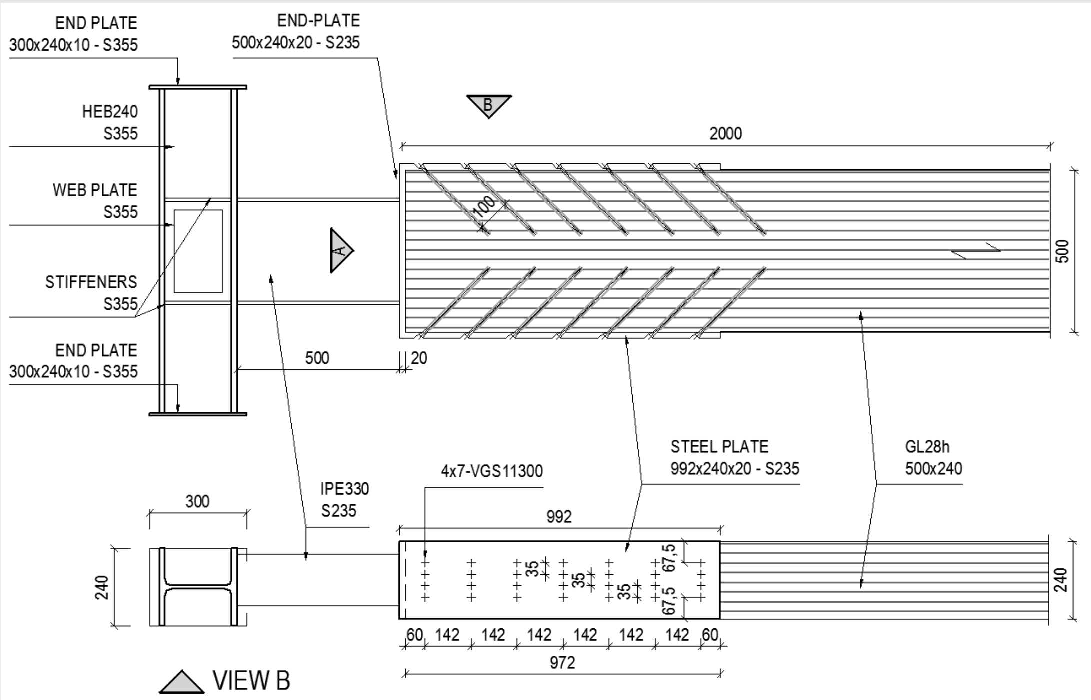


figure 11 – Edgewise Connection

configuration for the experimental campaign that is currently being developed at the University of Trento's laboratory.

3. FINITE ELEMENT MODELS FOR CALIBRATION

3.1. METHODOLOGY

GENERAL OVERVIEW

The necessity to predict the mechanical behavior of the proposed connection lead the research to the stage of performing a **numerical analysis** in Abaqus, a finite element software. The use of this computational tool enabled the visualization of the behavior of the structure under monotonic loading and gave the possibility to provide a failure mode, the stiffness of the connection and the value of overstrength offered by the connected elements to the steel profile, designed to be the dissipative element. After the results of the experimental campaign it is possible to evaluate if the prediction was satisfactory. The next step is the development of a finer calibration of the model and the assessment under cyclic loading.

In order to obtain reliable method for the modelling process, the first step is the validation of the system in a discretized way. The chosen models to **calibrate** the performance of the proposed technology was based on the available data from experimental tests performed on the same subject. The test campaign made in Trento for the connection with glued-in bars and the other one made for evaluation of the connection used for the timber dome for ENEL's Power Plant composed a valuable material to achieve this objective.

Besides the material strength, the main response of a structure is based in the system's resistance under **compression, tension and bending loads**. According to the experimental results these three elemental types of loading were covered in the validation step and the strength of the material could be adjusted accordingly and the damage evolution properties defined for each type of loading.

To be consistent with the results, the models followed the same method and assumptions. Following this concept, the input data is result of **calibrated parameters** based on experimental tests considering the same material. To define the input data that fits the best the failure modes, the maximum strength, stiffness and final displacements for each experiment in different load is a time consuming task. The lack of information on material properties is a problem especially for timber structures because of few tests dedicated to cover all the modelling issues, for example for fracture analysis. Another question to be raised is the high results scatter, considering that average values won't represent the worse situation.

It is important to mention that Abaqus has no built-in dimensions, hence the input values had to be kept in coherent **units**: forces were described in Newtons, the dimensions in millimeters and the stresses in MPa.

The process of validation of the tests started with the modelling of the bending test campaign performed test described in Andreolli's research on the **glued-in bars connection**. This model enabled the definition of the modelling parameters for steel elements including the influence on the results of damage evolution laws. For this case, the glulam beam was modelled as an isotropic material once no deformations are expected to happen in the embedment of the bars. This solution reduces the programming work and the development of a 3D model was possible to be developed.

The second group of tests was based on compression and bending tests performed at University of Karlsruhe and bending tests performed by University of Trento for the ENEL's Power plant dome. Since the modelling was considered valid for these three tests, the mechanical properties defined for all the tests and the fracture energy calibrated on the bending test were taken as input data for the proposed seismic link in bending once they act on the **same type of steel-to-timber connection**.

TYPE OF ANALYSIS

Situated in a paradox, this thesis is focused on the development of a simplified approach for the numerical analysis of a seismic joint but it is also dependent on complicated modelling issues because it is required to define elements with different materials and non-uniform meshes. To start the modelling process, first of all, it was defined which tools among the options offered by the FE package were more suitable for the stated problem. The usage of Abaqus/CAE tools was chosen to enable the modelling process to be done in a user-friendly interface.

To perform a **Dynamic Analysis** is the main objective of this research, hence an **Explicit Method** is the best solution to develop a computationally efficient process based on complex demands. With this type of analysis, the models can be developed with different material definitions and with non-uniform meshes, because it offers a fully automatic time incrementation process, integrating the equations of motion by an explicit central-difference integration rule together with the lumping of the element masses that can be performed even with a large number of small time increments. When compared to the direct-integration dynamic analysis performed in standard basis, it is a relatively **inexpensive approach**.

For **incremental-iterative processes**, Abaqus uses the modified Newton-Raphson method to recalculate the tangent stiffness matrix at every increment. In spite of this time saving approach, "for nonlinear problems the accuracy of the linear solution can impact the convergence of the Newton method", as decribed in the software Manual. Because of that, the iterative solver relative tolerance had to be manually specified in order to improve the convergence of the Newton-Raphson method.

Moreover, the process described above is the type of formulation that suits the most models that contain **shell elements** of S3/S3R and S4/S4R types. Since the use of such elements will be necessary for the development of the models in this thesis, making mandatory the choice of the described methodology.

This advantage is also helpful for models defining **nonlinearities** where the frequency of the structure changes continually hence its stability limit. When modelling materials, the nonlinear stress-strain relationships causes a nonlinear response of the material dependent on the time of the load application and on the load history and also environmental conditions. Condition that will be fundamental for the analysis of seismic resistant structures. To account for geometric nonlinearities caused by large displacements and rotations in a structure, Abaqus/Explicit uses a large-displacement formulation as default. This option was chosen for the developed models while the step was created.

The same type of analysis is also capable of defining a greatly **simplified interaction** definition with a general contact algorithm with small restrictions on involved type of surfaces in contact, adequate choice for the complex required models developed in this thesis. The contact was defined with tangential behavior based on a Penalty formulation of friction. The normal behavior of the defined contact was defined as hard contact for the pressure-overclosure allowing separation after contact.

The models were all **monotonically tested** by inputting a displacement to the timber beam. The solution to apply the load gradually uses an amplitude curve taken in a smooth step type through the whole running time of the analysis.

MODELLING STEEL

Defining material properties in a FEM requires different parameters depending on the type of analysis to be performed. As it was already mentioned, the **Dynamic Explicit** Analysis is the chosen method for the development of this thesis in order to obtain the structural behavior of complex and diverse structures.

Considering dynamic analysis, **density** is one parameter that should be specified for all materials. In case of steel, this value was taken as 7.85 E-9 ton/mm^3.

The **elastic behavior** is defined as isotropic through the values of 210GPa for Young's Modulus and 0.3 for Poisson's Ratio. The undamaged **plastic response** of the material is described in a true stress-strain relationship. While calibrating models it is important to input tabular values derived from experimental tests in order to extract a more close response from the FEM to the real system.

After inserting the mechanical properties, a smooth degradation in the material stiffness was applied in order to predict **progressive damage and failure**. This means that after a specified value for the onset of damage the stiffness matrix is successively modified in each step according to the specified damage evolution law. In the case of ductile metals, Abaqus provides a specific model that is suitable for dynamic situations. It is important to mention that the damage evolution response is set by default, in removing elements from the mesh if all its section points have lost their load-carrying capacity. By doing this the elements are progressively being "turned off" and the stiffness will be degraded. Despite the benefits of this model, there is mesh dependency of the results caused by strain localization effects during the progressive damage. The main objective for setting a damage evolution law, is to obtain a reliable modelling based on an equivalent degradation of the model in correspondence with the real response of the material. It affects specially the ultimate strain and the failure mode of the specimens, parameters defined by a post-elastic behavior.

Considering that no tests were carried out for the identification of the necessary data, the values used in this thesis were calibrated on the results from the experimental campaign developed for the glued-in steel bar connection. The tests resulted in failure modes based on the yielding of the steel elements: one specimen failure developed the yielding of the flange and another produced yielding of the flange combined with the yielding of the steel bars. For this reason, the different materials assigned for these two different elements offered the possibility to **calibrate the input** of the FEM by comparing load-slip curves. In a first step the steel plates and profiles were validated based on the specimen that failed in the flange and after the bars' material properties could be calibrated based on the second specimen. In order to describe the damage initiation it should be defined values for fracture strain, stress triaxiality and strain rate. The damage evolution is based on a displacement value at failure or on the fracture energy, which means the required energy for failure after the initiation of damage. It can be defined the descendant branch of the graph for the linear softening stress-strain response, hence the linear configuration is suitable for the most simple and default method.

It is worth mentioning that for the specimens in which the failure mode is localized in the timber element, the damage input values for steel materials didn't require efforts for validation. However, for the prediction of the proposed connection response, the validated values for damage initiation and evolution for steel elements described in this section were used as material property database for the proposed seismic link.

MODELLING TIMBER

Following the procedure described for modelling steel, modelling timber in dynamic analysis needs the specification of the same set of parameters: density, elastic behavior and damage initiation and evolution. However, timber is much more **complex** to be defined than steel and there are no proper tools in FE software to be applied. Specifically for modelling wood, there are researches working on the development of subroutines written for 3D solid elements with full integration to work in parallel with Abaqus, like for example the one made by Eng. Sandhaas. However, this is a long-term research and it should be considered that the progress in this kind of technology is still limited by the few available data to transcript the wood behavior in all the necessary parameters. Therefore, the development of a complicated model to simulate the real behavior of the material is unproductive in these present days. The judgment of such problematic is exactly as described by Sandhaas: "A first conclusion is to develop not too sophisticated and complex mathematical models. It is not effective to create overly precise models when the input data is sometimes inaccurate and sometimes just best guesses." (Sandhaas, Mechanical behaviour of timber joints with slotted-in steel plates, 2012).

Based on this conclusions, instead of engaging in such an expensive study as codifying subroutines, which is highly dependent on laboratory tests for the validation process, this thesis used a **simplified method** to model wood based on the available tools in the FE package that can be applied for the case under study.

In a primary discussion, it should be emphasized that wood is a **natural material** made of an arrangement of cells in one oriented direction, which is result of the development and growth of the wood tissue (Kuklík, 2008). The composition of cells in a trunk varies from tree to tree depending on environmental conditions during the growth period, obviously on the type of specie and on the presence of knots. This non-homogeneity is the cause of high value scattering while defining mechanical properties in timber elements.

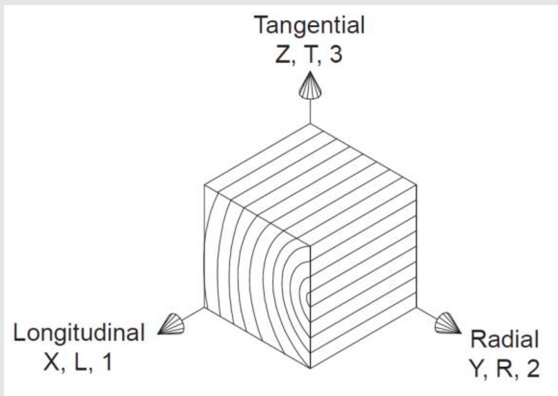


figure 12 - Definition of material directions

Furthermore wood is an **anisotropic material**, what means that its properties are different depending on the considered direction. This characteristic can be arranged in three principal axes as Cartesian axes of reference: longitudinal, radial and tangential as shown in figure 12. However, the simplification defined in standards states wood as transversally isotropic accounting for only two directions: parallel-to-grain and perpendicular-to-grain. This reduction fails to represent the real behavior of wood because there is a great difference between the characteristics in transversal and radial directions. The elastic modulus in tangential direction is about half of the radial one. Therefore, it is important to define an orthotropic material in mathematical models in order to obtain a behavior more close to the real.

After this statement, it is important to point out the problems involving the **reliability of values** for defining properties for wooden elements. For example, when inputting values to define wood as orthotropic material it is often necessary to quest for data in the available literature because it is difficult to obtain the high number of tests results needed to explore the subject. Adding to this it should be also mentioned that the tensile and compressive yield stresses should be determined using pure uniaxial loading tests (Hong, 2007), which is difficult to obtain without having secondary stress components (Sandhaas, 2012). Another variation involves the size of the tested specimen because material properties are highly influenced by grain angle and mainly by the presence of knots.

When dealing with **combined glulam**, it should be highlighted that one structural element is result of a random distribution of segments with different material properties causing high variance of quality inside the same member (Melzerová, Kuklik, & Šejnoha, 2012). However this questions cannot be taken into account without developing a complex model based on probabilistic simulations.

Focusing on defining a methodology for studying the case of the proposed connection in practical applications, Abaqus package offers the option of defining an **orthotropic material** in instantaneous time scale for the transcription of elastic mechanical properties,. The input data has to be arranged in a matrix 'D' as described in a set of formulae presented in equation 6, involving modulus of elasticity, shear modulus and Poisson's ratios for three directions.

$$
\begin{aligned}
&Y := \frac{1}{1-\nu_{LR}\cdot\nu_{RL}-\nu_{RT}\cdot\nu_{TR}-\nu_{TL}\cdot\nu_{LT}-2\cdot\nu_{RL}\cdot\nu_{TR}\cdot\nu_{LT}}\\
&D_{1111} := E_L\cdot\left(1-\nu_{RT}\cdot\nu_{TR}\right)Y\\
&D_{2222} := E_R\cdot\left(1-\nu_{TL}\cdot\nu_{LT}\right)Y\\
&D_{3333} := E_T\cdot\left(1-\nu_{LR}\cdot\nu_{RL}\right)Y\\
&D_{1122} := E_L\cdot\left(\nu_{RL}+\nu_{TL}\cdot\nu_{RT}\right)\cdot Y\\
&D_{1133} := E_L\cdot\left(\nu_{TL}+\nu_{RL}\cdot\nu_{TR}\right)\cdot Y\\
&D_{2233} := E_R\cdot\left(\nu_{TR}+\nu_{LR}\cdot\nu_{TL}\right)\cdot Y\\
&D_{1212} := G_{LR}\\
&D_{1313} := G_{LT}\\
&D_{2323} := G_{RT}\\
&D := \begin{pmatrix} D_{1111} & D_{1122} & D_{2222} & D_{1133} & D_{2233} & D_{3333} & D_{1212} & D_{1313} & D_{2323} \end{pmatrix}
\end{aligned}
\qquad \textit{equation 6}
$$

Specifically for the **validation models** in which GL28c is specified, the stiffness values were derived from the bending and tension test results from the experimental campaign developed for the characterization of the ENEL's dome. In order to fulfil all the necessary data to obtain a 3D behavior relatively close to the tests, the missing data was obtained from common values for wood. Values for Poisson's ratio were derived from the average number for conifers and through the formulae presented by Piazza (Piazza, Tomasi, & Modena, 2005) and described here in a set of equations labelled as equation 7.

$$
E_T := \frac{E_R}{1.6} \qquad
\begin{aligned}
\nu_{RL} &:= \frac{\nu_{LR}}{E_L}\cdot E_R\\
\nu_{TL} &:= \frac{\nu_{LT}}{E_L}\cdot E_T\\
\nu_{TR} &:= \frac{\nu_{RT}}{E_R}\cdot E_T
\end{aligned}
\qquad
\begin{aligned}
G_{LT} &:= 0.94\cdot G_{LR}\\
G_{RT} &:= \frac{G_{g.mean}}{10}
\end{aligned}
\qquad \textit{equation 7}
$$

The **density** was assumed with an average value of 442 kg/m^3 presented among the test results for the dome connection. Comparing with the standard value of 380 kg/m^3, this is even higher value than the one defined for homogeneous GL28. It is known that this feature depends on the moisture content. However, the focus of the research is on instantaneous actions, what allows the statement that the climatic conditions don't change during the seismic event. It is considered temperature of 20 ºC and relative humidity of 65% same reference as for service class 1 in EN 5 in which the average moisture content doesn't exceed 12%.

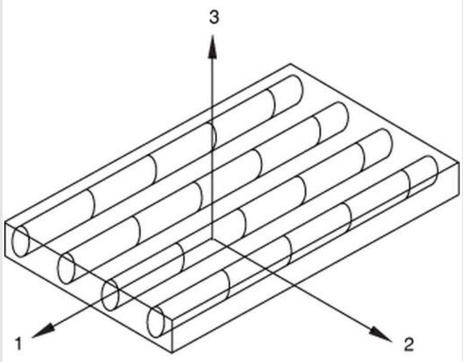


figure 13 – Unidirectional lamina (from Abaqus Manual)

To account for the complex behavior of timber that is ductile in compression and pseudo-brittle in tension the model needed to define a stress-based failure envelope like the one defined by Tsai-Hill quadratic surface based on maximum stresses presented in figure 14. Especially for this case of study, the selected model, among the options offered by Abaqus software was the **Hashin damage**, a damage model specially developed to predict anisotropic damage in elastic-brittle materials. Wood composition is not very far from the composition of fiber reinforced materials with aligned fibers. The necessary input data to define the onset of failure is defined in Standards or in common results from experimental tests, they are: longitudinal tensile strength, longitudinal compressive strength, transverse tensile strength, transverse compressive strength, longitudinal shear strength with exception of the transverse shear strength, which is calculated automatically in Hashin theory proposed in 1980 as being half of the transverse compressive strength. By doing that, the values are higher and this effect is disregarded, what is reasonable for the current application because the rolling shear failure is not usual to happen. Another advantage of this mode is that it keeps the material undamaged during the elastic analysis, perfectly suitable to represent the wood behavior.

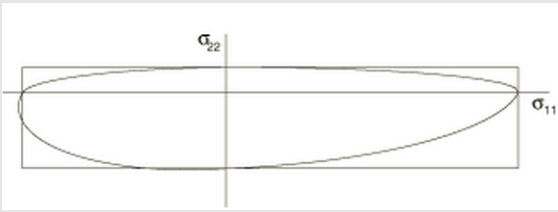


figure 14 – Tsai-Hill maxim stress failure envelope (from Abaqus Manua)l

This type of damage can be used in combination with the **damage evolution** model based on the energy dissipation during the damage process and linear material softening. Element failure models allow elements that reach high strains to be removed from the model. This option is useful to adjust the ductile behavior of wood in compression and to obtain the adequate failure model of the structure. The needed input includes fracture energy of the lamina in the longitudinal tensile and compressive direction and the transverse tensile and compressive direction.

The disadvantage of this model is that it can just be applied to **plane stress continuum elements**, like shells for example. The suppression of the third dimension in damage reduces number of input variables, transforming the material properties in this direction just a stabilizing feature, because its stiffness influences in the stable time increment size.

To enable the use of this damage model, the solid elements had to be separated in **longitudinal-radial "slices"** with and without the screws row. The shells were tied together with no significant difference in the behavior of one single layer. Another measure that was implied to the modelling process was the compatibility of meshes to accelerate the integration and also part of an effort to reduce mesh dependency of such type of models by keeping the same size of elements for the mesh even when dealing with different geometries, in order to have a correlation between the energy and the different validation models.

The parameters necessary for characterizing the mechanical behavior of the assessed structures were calibrated on the **experimental results** produced for the ENEL'S dome joint. The compression strength was calibrated based on the load-displacement curves obtained from the compression tests performed by Karlsruhe University. From the same laboratory, the tension test results were used to calibrate the tensile strength and the modulus of elasticity was reduced to consider the connection slip. Finally, the test carried out by University of Trento was useful to calibrate the fracture energy related to the material and failure mode of the structure subjected to bending.

The values of **fracture energy** were derived from each test: compression, tension and bending results following the principal that if the material is more brittle lower is the value to be defined. In one determined combination of values, the failure mode is achieved and the model is considered calibrated. Since the mechanical behavior of the timber can be classified as elastic-plastic under compression and in tension by linear softening behavior, the results were oriented to obtain this final result.

Despite of the good correlation of results between the FEM and the experimental data, to have a full calibrated model, experimental tests in single elements should be performed in order to investigate exclusively the mechanical properties of the each used material separately.

The following sections present the three validation models made in order to obtain the timber properties used in the dome connection. These values are used as input for the following step to predict the overstrength of the proposed connection in relation to the seismic steel link.

3.2. STEEL LINK IN BENDING

EXPERIMENTAL DATA

The component model described in this section was developed based on the experimental campaign made in **University of Trento**, which was described in paragraph 2.2 - DESIGN OF CONNECTIONS. The specimens chosen for the modelling calibrations were the P06 and P10 that presented the most representative ductile failure modes from the monotonic loading tests. The test setup creates a stiff and rigid

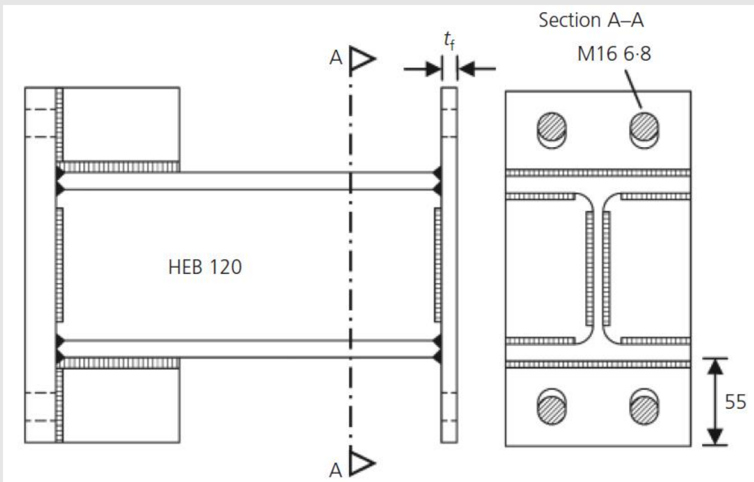


figure 15 – Details of the glued-in bars connection (Andreolli, Piazza, Tomasi, & Zandonini, 2011)

connection in one end of the profile and in the other the different thickness plates are welded and connected by glued-in bars to the timber beam in a cantilever configuration.

The timber beams had a rectangular cross-section with 120 x 240 mm^2 in GL24h. The steel components were made in steel grade S235 including a HEB 120 profile welded end-plates (by full penetrations butt weld). Among the tested thickness of the end-plates, 6 and 10 mm were taken for the numerical analysis. The four bars were M16 6.8 placed inside slotted holes in the end-plate and carefully glued centralized in 18 mm diameter predrilled holes in the timber beam.

The results were presented in a moment-rotation capacity history curve in order to assess the behavior of the connection and the influence of the different failure modes on the ultimate rotation.

MODELLING PARAMETERS

The geometric parameters described above were reproduced in different **parts** modelled in a solid Abaqus/CAE and assembled in its proper positions. Washers, nuts, stiffeners and slotted holes in the end-plate were considered. The full penetration butt weld allowed the profile and end-plates to be modelled as one single part without the weld geometry simplifying the computation process time by reducing the number of interactions and different kinds of contact needed.

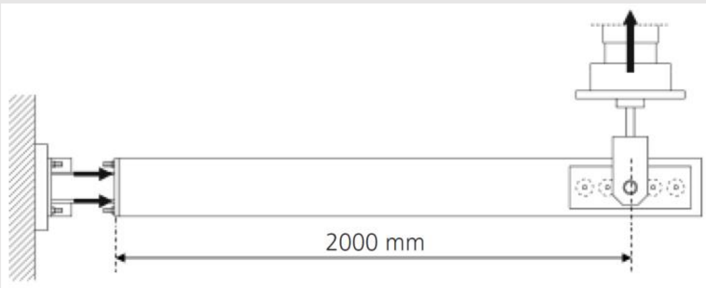


figure 16 - Details of the set-up test for monotonic loading (Andreolli, Piazza, Tomasi, & Zandonini, 2011)

Fixed **boundary conditions** were imposed to the free end-plate. The free end of the timber beam starts from a steady-state condition and is subjected to a controlled displacement applied smoothly through the whole step time.

Since there was no relative displacement observed between the bars and the timber beam during the experimental campaign, the embedment contact was reproduced as **tie**. The holes in the beam were considered with the same diameter and length as the bars and the master surface of the glulam was constrained to the slave surface of the bars restraining the relative movement of the different elements. This modelling decision simplified the analysis by excluding from the modelling issues involving the properties and behavior of the glue, as well as its interaction with the connected elements.

The Abaqus/Explicit analysis allows the use of the **general contact** between the parts with only one global contact properties. This simple and fast interaction option that was described in the previous section was defined by a general tangential and normal behavior. For this model it was considered the penalty based formulation with a friction coefficient of 0.5, value taken by Sandhaas et al. in 2012 while the modelling of a dowel embedded in wood (Sandhaas, Kuilen, & Blass, Constitutive Model for wood based Continuum Damage Mechanics, 2012).

The **glulam** was defined like an isotropic material because of the observed tested specimens, in which the beams acted like rigid bodies and presented no failure mode related to the stresses perpendicular to the fibers in contact with the bars. The input material properties were the mean value of modulus of elasticity parallel to grain, the Poisson's ratio for the radial-transversal plane, the characteristic density and the characteristic bending strength, values taken from EN 1194: 2000 for the specified glulam.

For the plates and profile, the **steel** properties were taken by the available experimental data (Andreolli, Piazza, Tomasi, & Zandonini, 2011). The published paper presents the measured the ultimate strain and yield and ultimate strength of each end-plate in S235, differing the thickness. Hence, these values were used as input data for the definition of the plastic mechanical properties. The general elastic properties were considered 210 GPa for the Young's Modulus and 0.3 for Poisson's ratio. The onset of failure and damage evolution were defined by the software available formulation specific for ductile metals. The needed input values for ductile damage initiation, fracture strain, stress triaxiality and strain rate were calibrated on the tests results.

In order to keep the geometry of the experimental tests, the **bars** were modelled with their nominal diameter. Hence, the input material properties were altered to consider the reduction of area caused by threads. The effective area transformed the value of affected parameters: the elastic modulus and plastic mechanical properties. Since there were no experimental tests performed for the bars, the yield and ultimate strength were taken following the class 6.8 and the ultimate strain as 0.14. The damage was input in the same way as described for steel plates: through damage for ductile metals and the necessary parameters were assessed by calibration based on the experimental results.

Mesh patterns influence the time of the analysis and visual failure modes, consequently it is important to simplify and obtain the most regular and uniform mesh as possible. The critical part for mesh compatibility of the studied models is the curved geometry of steel hot rolled profiles. In order to reduce the creation of elements that are not clearly solved by the software and can result in wrong stiffness interpretations, the curved elements were transformed in an equivalent triangular shape. Another strategy used to have a more efficient analysis was the definition of fine mesh in zones where deformations were expected and coarse mesh for the stiff elements. For example the flanges and end-plate were divided in smaller elements than the timber beam.

ANALYSIS OF RESULTS AND COMPARISON

In order to have comparable values, the right directions and position of the results matter. To extract from the numerical model the same answers given by the experimental analysis, the rotation capacity and the bending moment were obtained from the same node coupled to the timber surface, around 2 meters distant from the application of the forced displacement. By means of a **history output request**, the rotation-time curve of the mentioned node was plotted and the load-displacement curve was plotted for the point of load application. With this information, the moment was calculated from the force and the distance to the rotational controlled node.

As it was mentioned before about fracture mechanics, the input data for the **damage initiation and evolution** were obtained by calibrating values when comparing the numerical and the experimental results. The specimen P6 presented failure based on the complete yielding of the flange, therefore its results were used for the calibration of the steel material definitions. Since the specimen P10 presented failure dependent on the bar failure and flange yielding in the presence of prying forces the bars' material input was calibrated after the validated steel material. The damage evolution is an important parameter to provide the strain hardening and the ultimate rotation capacity of the models. It is associated with the correct prediction of the failure mode and the ultimate resistance (Andreolli, Piazza, Tomasi, & Zandonini, 2011).

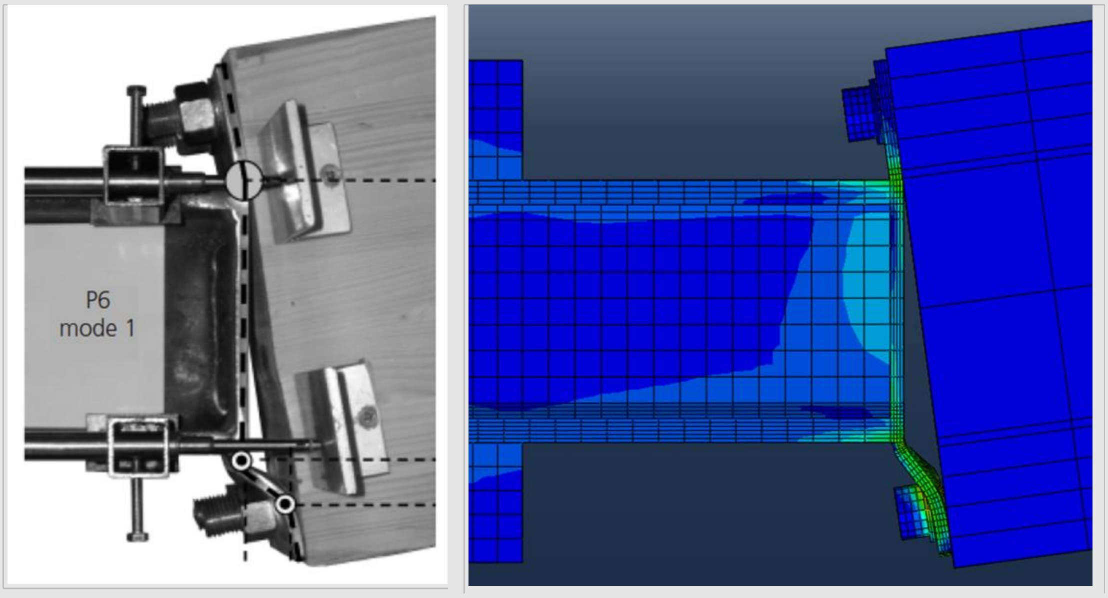


figure 17 – P6 Failure Mode - Test (Andreolli, et al., 2011) and Abaqus results

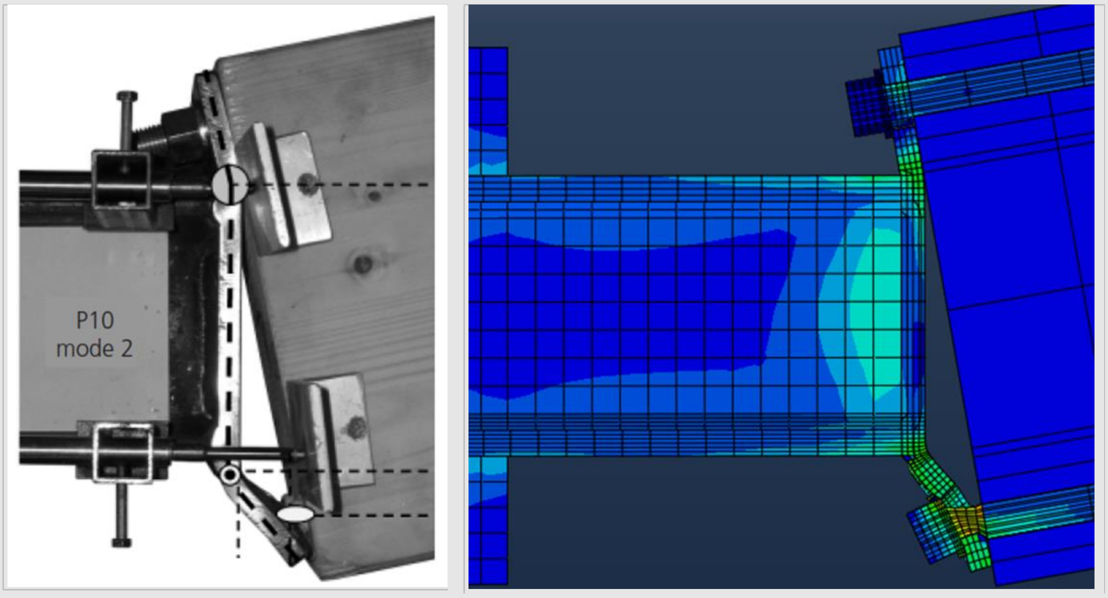


figure 18 - P10 Failure Mode - Test (Andreolli, et al., 2011) and Abaqus results

Finally, after all the calibration efforts, the models demonstrated consistent results with the same input data but different geometry, varying the thickness of the end-plate and the length of the glued-in bars. When comparing the **moment-rotation capacity curve** of the real test with the numerical one (figure 19), it is possible to observe a good equivalence of the outputs for initial stiffness and rotation capacity, qualifying this FEM for the characterization of this joint in the same connection technology. The difference in moment capacity of the model with end-plate 10 mm thick is due to the hardening property of the bars, which was not considered in the analysis. Once the objective of this model was the quality of the results for the steel link S235, the calibration process was considered efficient.

It is important to emphasize that the obtained failure modes presented in figure 17 and figure 18 are clearly close to the real one, what assure the model's capability to reproduce the real behavior of the structure.

Lastly, this FEM here developed can be useful for the **prediction of the global ultimate resistance** and rotation of the same test configuration, enabling the optimization of the solution and further development of a series of parametric studies.

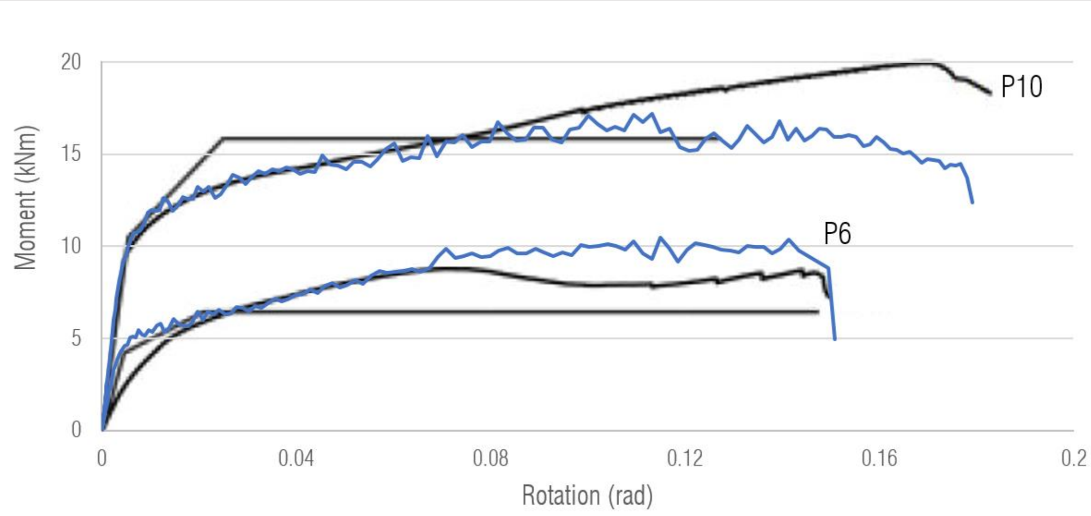


figure 19 - P6 and P10 Moment–rotation relationship: experimental results compared with the trilinear theoretical curves (Andreolli, Piazza. Tomasi. & Zandonini. 2011) and with Abaqus results (in yellow)

3.3. GLULAM IN COMPRESSION

EXPERIMENTAL DATA

The compression test developed for the ENEL's dome was performed by the **University of Karlsruhe** (Germany) in the Karlruher Institut für Technologie, VAKA under supervision of Univ. Prof. Dr. Ing. Hans Joachim Blaß. The result values were described in a report presented on November, 2012, which were used for the validation of the numerical model under compression parallel to the fibers.

The **test set-up** reproduced one node connection: a GL28c beam with 180 x 1132 mm^2 and 1150 mm long was placed vertically and loaded centrically on its top against two steel plates of 120 x 520 mm^2 placed as supports and making a space in the middle for the embedment of the vertical shear plates grooved in a small depth of the beam. Under the plates the node steel elements were not assessed in this test while the timber is the governing failure mode.

In more details the timber beam has a **chamfer** finishing for the connection with the bottom plates reducing its width from 180 mm to 120 mm. Moreover, the timber fibers were considered to have an **angle of 4.7 º to the vertical axis**, consequently to the direction of load application.

The tests results were characterized by **load-displacement curves** of 5 specimens that was used as base for calibration of the numerical models.

MODELLING PARAMETERS

Once the tests were made for the assessment of the glulam under compression loading, only the **timber beam** was modelled, reducing issues of interaction between elements and integration time.

In order to develop the model based on the timber behavior, **continuum and homogeneous shell** parts were created. To simplify the model and still consider the reduction in thickness of the chamfered part, the geometry of the test was reproduced and different thickness were assigned to the model but in a step, changing 180 mm to 120 mm at once. The reduced area of the support was also considered, because this was one of the causes of failure in the real tests. Hence, the **boundary conditions** reproduced the test by fixing the bottom edge of the continuous shell just in the length of the two plates. The load was applied as a controlled displacement on a single node coupled to the whole top surface of the specimen. The area of distribution of the load was assumed as the entire top section because there was no precise description of the loading conditions in the test reports.

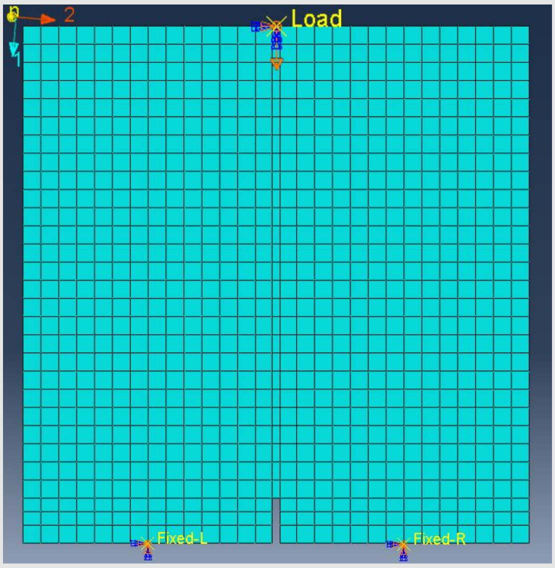


figure 20 - Compression test - Material orientation and boundary conditions

The input **material properties** were defined according to the mean values presented in the test reports. The use of the values described in the code EN 1194 transforms the models outputs in incomparable values because the real beam was assessed with characteristic compressive strength of 34 MPa instead of 24 MPa, the standard value for GL28c. The density was also taken from the experimental results. The rest of variables were taken from standards for the specified glulam.

For orthotropic materials, it is necessary to assign orientation to the material properties. Considering that the beams were produced with **inclined fibers**, an angle of 4.7º was created from the vertical axis to the longitudinal direction of the wood fibers. This number was taken from the design drawings of the dome nodes and the rotated rectangular coordinate system can be seen on the top of figure 20 (the axis 1 is associated with the longitudinal direction and 2 with the radial).

The meshing pattern was controlled by in a quad dominated structured technique with **S4R elements**: "a 4-node doubly thick shell, reduced integration, hourglass control, finite membrane strains", suitable for general purposes and applications.

ANALYSIS OF RESULTS AND COMPARISON

The expected failure mode of timber elements loaded in compression parallel to grain is associated to the instability of the fibers and local buckling of the cell walls. The onset of damage starts when one fiber buckle and establishes the end of the elastic phase. The plastic behavior is characterized by an increasing deformation at an almost constant load, while there is a propagation of crumpling and buckling of the fibers. This behavior can be seen in the failure mode presented in figure 21.

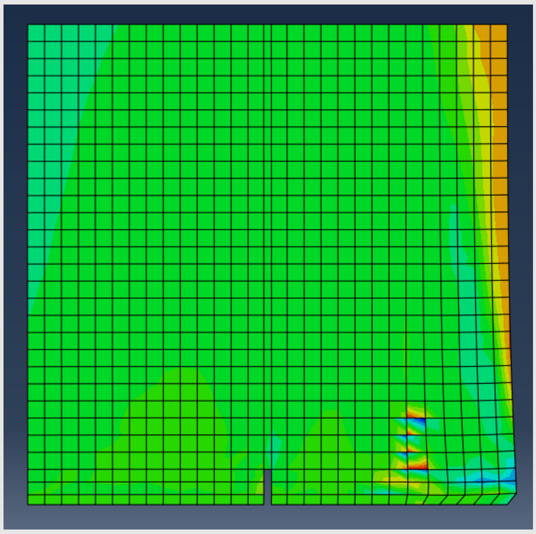


figure 21 - Compression test – Abaqus failure mode

The failure modes of the experimental tests showed compression and tension in perpendicular direction, same response gave by Abaqus model.

In order to have an input value coherent to the material properties of the used glulam, the compressive strength was taken as the average of the values found in the test reports. Hence, it was possible to achieve the maximum applied force inside the experimental results scatter. The ductile behavior in the model could be achieved with appropriate damage evolution data that was calibrated on the tests results (see figure 22). With this calibration is also possible achieve a maximum displacement comparable to the relative values described in the test report.

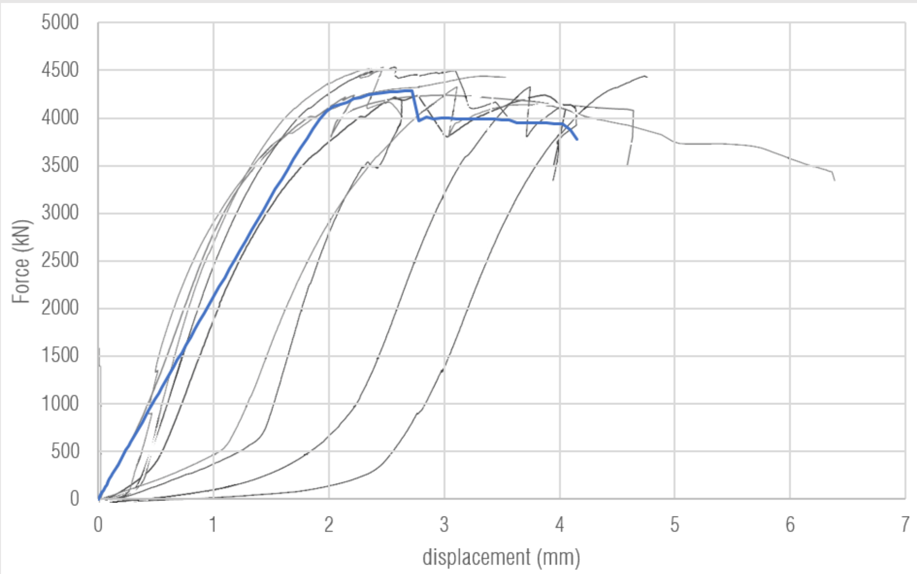


figure 22 - Compression test – Abaqus output compared with the range of results from the experimental tests

3.4. GLULAM AND CONNECTION IN TENSION

EXPERIMENTAL DATA

The experimental data used for this component test was also performed in **University of Karlsruhe** part of the same campaign of mechanical characterization of ENEL's dome.

The **test setup** was produced 10 specimens with a GL28c beam with 566 x 180 mm^2 (half of the real depth) 3 m long and steel plates made from S355 with width of 180 mm, thickness of 20 mm and length of 795 mm plus 300 mm for the connection with the machine support for loading. The plates were placed for an edgewise connection in both ends of the beam with screws connected to both ends. Each plate was predrilled for the insertion of 24 screws, in total, there were two plates and 48 fasteners per specimen. The full threaded self-tapping screws were RothoBlaas VGS Φ11 x 450 mm and were placed at an angle of 45º.

Through the edgewise plates, the specimens were loaded in tension and the **load-displacement curve** was plotted.

MODELLING PARAMETERS

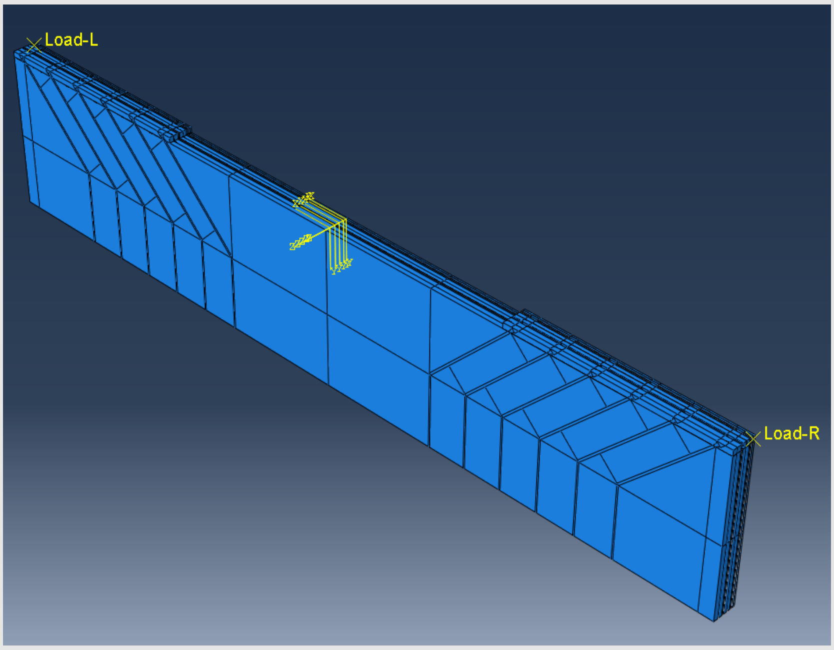


figure 23 - Tension test - Assembly

This component model is more **complex** than the compression test because it includes different material and interaction between parts. It is difficult to distinguish each elements contribution for the response of the structure to the load.

The first issue to be solved is how to simplify the 3D solid elements in continuum homogeneous shells in order to use the damage model given by Hashin Damage used just for the timber members. As it was mentioned in section 2.2, the load bearing capacity formulated by the European Standard considers the contribution of the embedment in timber only along the diameter of the screw, moreover, in case of full threaded screws, it should be considered an effective value of this diameter. Therefore to characterize such a connection, the beam and plates were "sliced" into **longitudinal-radial planes**. Four planes with the definition of the screws and five planes with entire dimensions for the plates and beam (see figure 23). The outer shells are important to provide the right stiffness to the model. By just considering one slice of the structure, the model cannot reach an equivalent strength when compared to the test results.

The “confined effect” provided by the external layers are fundamental for the development of a response comparable to the real one.

The **geometry** of screws was considered in a uniform squared cross-section with effective diameter as dimension side. The length was considered shorter in one nominal diameter in order exclude the point part as proposed by EN 5.

For the assignment of interactions, **tying constrains** for all directions were modelled between the shells based on the fact that there was no failure related to the embedment region of the fasteners in any specimen. The interaction properties were assumed to be equal between all the surfaces by means of a general contact with the same parameters as described previously for the bending model with glued-in bars.

The load was applied on the end of the plates by imposing **a controlled displacement** with smooth amplitude definitions. In order to obtain a load-slip history of the analysis, a reference node was coupled to the movement of the loaded surface.

By means of **boundary conditions**, the model was simplified to one connection on half of the thickness, which means five parallel planes for the shells: two with screws, one outer layer, one middle layer and the in-between row of screws. This solution reduced the integration time and made the modelling process more efficient and no variation in the mechanical behavior was noticed.

The **mesh** map was defined in order to be the most uniform with less distorted shapes as possible to follow the geometry of the structure. The inclination of the screws created a complex region that needed to be divided into simple elements by a minimum number of partitions. By doing that the elements integrations are faster the results are more stable. The mesh controls were defined as quad-dominated free type to be possible to match triangular and quads. The regions in which it was expected to concentrate stresses and deformations were finer seeded in order the behavior of the structure in a more detailed way. For the mesh module, the explicit element library the element more suitable for the analysis are the S4R, already described in the section 3.3 and the S3R: “a 3-node triangular thick shell, finite membrane strain”.

Finally, the **material properties** followed the methodology described in section 3.1 and calibrated with strength values higher than the classification of the glulam limits. The elastic modulus of elasticity was reduced in a calibration process based on the required displacement on this test in which connection slip can be observed. This reduction also fits the moment-displacement curve presented in the test reports for the bending test. The damage evolution was strictly calibrated for this model with the failure mode based on the rupture of the timber fibers.

For modelling the material of the plates, the information obtained from the experimental campaign in tension which produced a valuable data to set the real ultimate strength of the used steel.

ANALYSIS OF RESULTS AND COMPARISON

During the experimental campaign all the specimens presented governing failure mechanism based on the block shear of the glulam beams. In the orthogonal direction, cracks could be observed at the point of the screws because of localized tension in this regions. The presence of this type of cracks is acceptable when the stresses can continue be developed while crack growth converges.

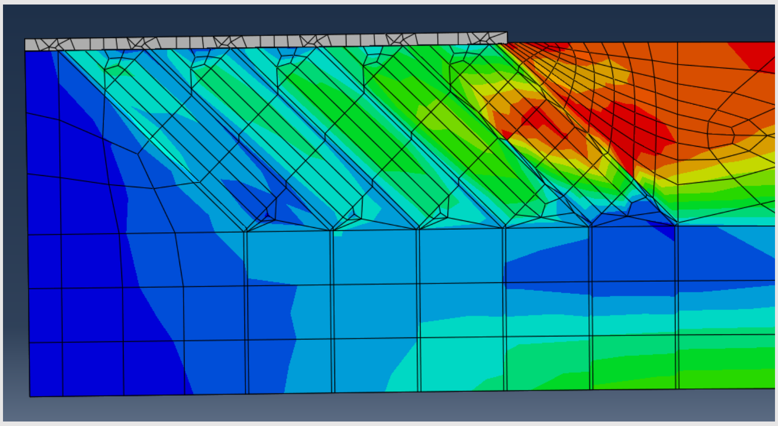


figure 24 – Tension test – Abaqus failure mode

Since the failure starts localized in a region close to the end of the connected plate, there was no occurrence of failure due to defects in the outer lamellae of the specimens. Hence, the prediction of failure in a numerical model can be disconnected of a random location of failure because of a weak part of the element, due to knots for example. This location of failure could be reproduced by Abaqus by observing the defined failure mode in figure 24.

Because of the complex system defined by the inclined screws and the asymmetry created by the load, there is no straightforward method to define clearly the distribution of stresses from this test, therefore it is difficult to assess the real tensile strength of the material.

In spite of that some assumptions were made based on the observation of the tests results. The compression tests indicate a glulam stronger than the specified grade, as well as the density data from both types of loading. Therefore, the properties of tensile strength were defined as being higher than the standard value and calibrated on the mean values presented in the tests results for maximum force and displacement. The quasi-brittle behavior of timber could be well represented by the model and the good correlation with the results can be seen in figure 25.

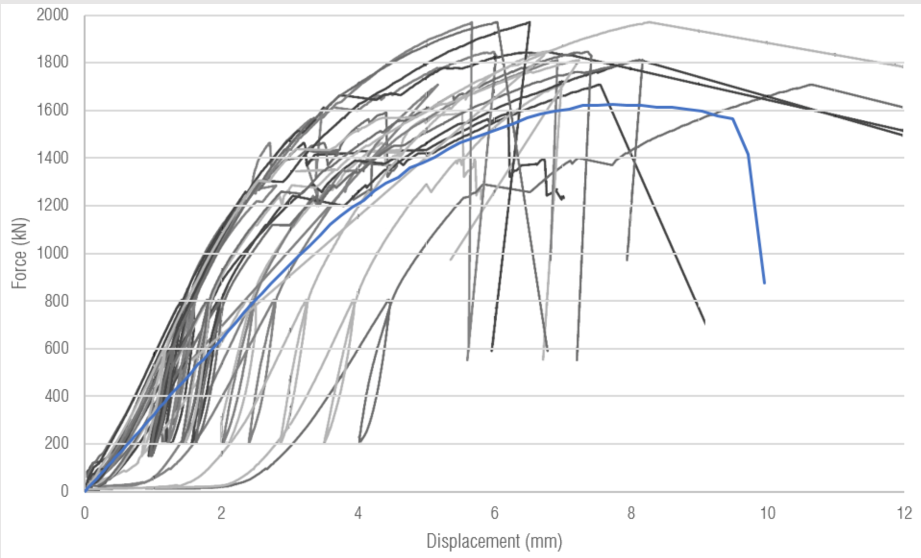


figure 25 – Tension test – Abaqus output compared with the range of results from the experimental tests

3.5. GLULAM AND CONNECTION IN BENDING

EXPERIMENTAL DATA

The tests for bending of ENEL's dome node were carried out by **University of Trento** (Italy), in the Materials and Structural Testing Laboratory, Timbertech under the supervision of Univ. Prof. – Ing. Maurizio Piazza. The description of the results was also presented in the same report published on November of 2012 (figure 26).

The **test setup** had different load configurations but the one based on a 3 point bending was used for the modelling in order to assess the connection in the best way. Two timber beams were connected to the steel node by the edgewise plates with screws. The plates and screw configuration is the same as the presented previously for the tension experiment. The length of each timber beam is 4.5 m producing a total specimen with more than 9 m long.

figure 26 - ENEL's Dome - 4-point bending test realized at University of Trento (Mazzolani's photography)

One special condition of the test is that the **beam axis was rotated** from the horizontal axis in 4.7º. The structure was simply supported and the load applied on the node. Different from the previous experiments, the test report presents a load-slip curve, giving more information for calibration processes.

It is important to mention that this test was just carried out twice.

MODELLING PARAMETERS

The modelling procedure followed the same concept of the developed for the tension test. The geometry of the beam and connection suffered changes but the parts were created with **continuum homogenous shells**.

The specific alterations of the tension to the bending model rely on the **inclination of the beam** and that the edgewise plates were linked to a vertical endplate producing a "C" shape for the connection with the dome node. This element represents the support system for the compression forces developed on the top part of the cross-section of the timber beam.

The **boundary conditions** had to be complemented by the end supports. No simplification to develop a model in half beam height could be done as in the tension test, because the test has no such symmetry. However, it could be kept the reduction in half thickness of the system in order to decrease the computational time.

Related to the mesh pattern, material definitions and definition of interactions between the different elements, all the modelling procedure followed the one developed for the tension one. The load was also applied as a controlled displacement and the material properties being determined by the result of a calibration process of the three component models.

The stiffness of the model was calibrated on the tests results and the strength defined for damage initiation was based on the standard values for the specified combined glulam in order not to consider the higher strength of the real specimens as it was defined for the tension test and compression. This was decided aiming not to produce an overestimation of the material mechanical properties focusing on the next step of the FEM: investigating the proposed connection behavior with the parameters defined with the dome bending model.

ANALYSIS OF RESULTS AND COMPARISON

From the experimental campaign report, the given curves from the moment-displacement for the 3 point test were considered to assess the stiffness of the glulam used in the dome, which values should be lower than the standard ones in order to account for slippage of the connection.

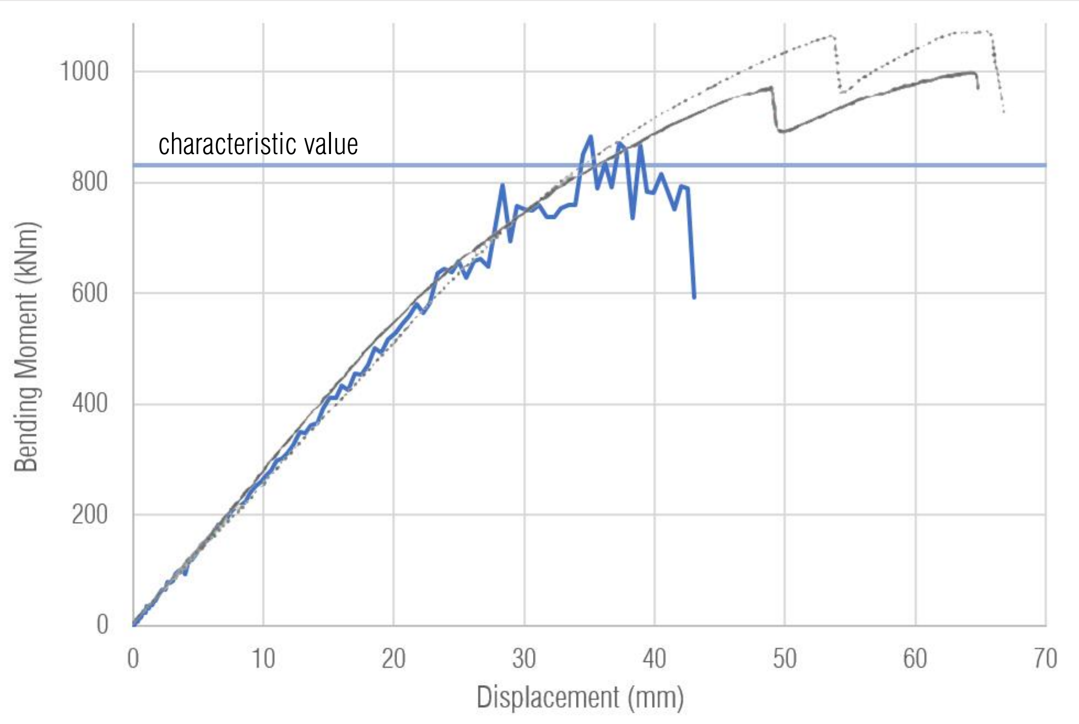


figure 27 - 3 Point Bending test - Abaqus output compared with the characteristic value for moment and curves defined in the experimental report

Another possible reason for altering the modulus of elasticity is due to size-effect, a full-scale specimen can differ from the standard values for mechanical properties of timber. It is known that modulus of elasticity might be affected by defects in each lamination and also reflect a deficient process of the grading criteria for classifying the lamellae. This last reason was described in the experiments performed by Sousa in 2013 while assessing the mechanical properties of glulam, classified as GL28h by the manufacturer, but that presented in reality strength and density consistent with the standards requirements and stiffness properties lower than it could be acceptable for such timber grade (Sousa, Branco, & Lourenço, 2013).

By observing the output graph presented in figure 27, it is clear that the post elastic behavior doesn't reproduce the same shape as the real test. However the model was calibrated in order to fulfill the characteristic value for the moment defined in the experimental report considering other bending tests, for example 4 point bending tests. The calibration was considered successful once it could achieve this characteristic value and the right displacement. The rupture of the fibers occurs in a first place in the end of the edgewise plate and the stresses are distributed reaching the plane formed by the points of the inclined screws. After that, there is propagation of cracks and it can be observed in the splitting effect, delamination the glulam (see figure 28).

The bending failure mechanism achieved is in accordance with the common failure mechanism described by Souza et al. as "the failure of the lower fibbers by tension and creation of horizontal cracks in the proximity of the neutral axis" (Sousa, Branco, & Lourenço, 2013).

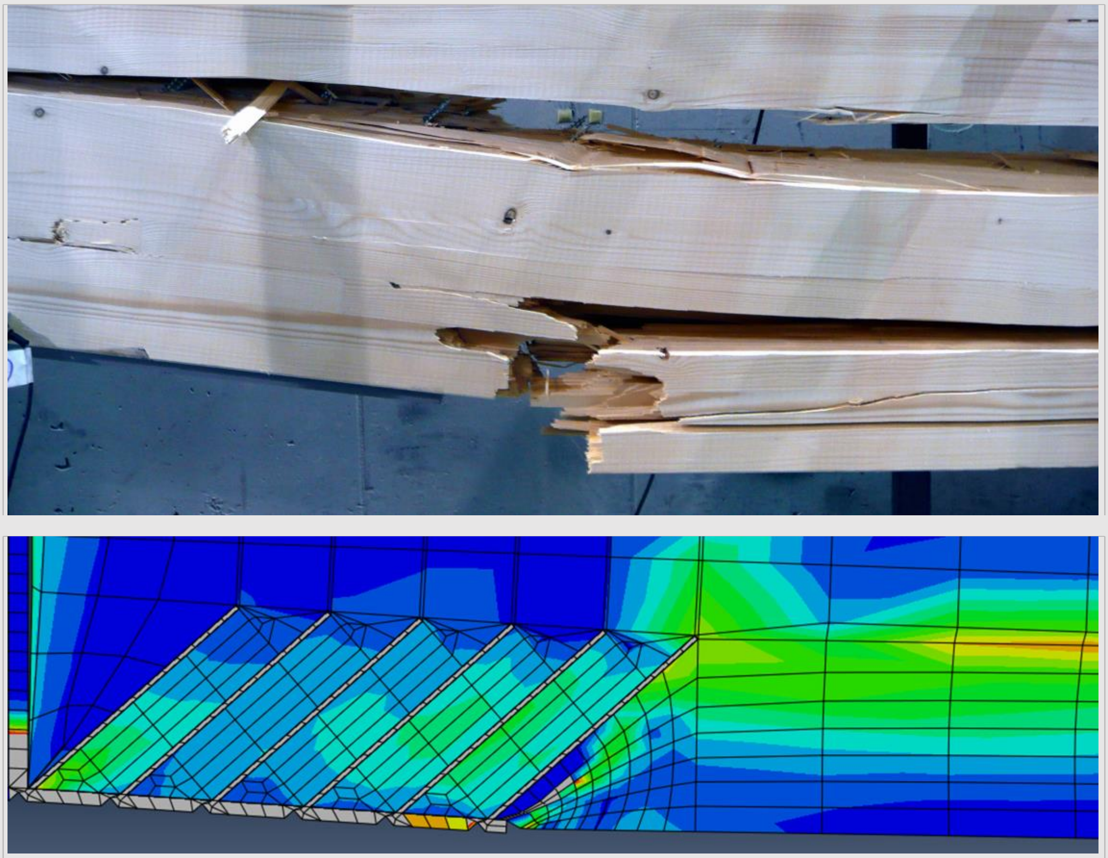


figure 28 – Bending Test – Comparison of experimental test rupture (Mazzolani's photography) with Abaqus output

4. FINITE ELEMENT MODEL OF THE PROPOSED LINK

4.1. MODELLING PARAMETERS

GENERAL OVERVIEW

Recalling what was already mentioned, the main objective of this thesis is to develop a FEM willing to predict the behavior of a seismic steel link inserted in a timber beam by inclined screws in edgewise connections. The study case presented in section 2.2 relies on the ductile behavior of the steel profile to deform inelastically keeping the connection and timber element in elastic response. Hence, the numerical model of this joint was modelled in Abaqus to verify the overstrength of the elements in relation to the link.

Following the procedure described in section 3 for modelling all the involved elements, three models were performed to complete the necessary verification. The first FEM to be developed was based on solid elements aiming to assess the behavior of the steel link when subjected to monotonic load. The moment-rotation capacity can be predicted based on the calibration data obtained from section 3.2. The second model and the third model had the same geometry as the former but were modelled with shell elements in order to obtain the response of the connection: plate-screw-timber and of the timber beam alone. Both resulted in a maximum bending moment that could be compared to the value extracted from the first model.

The geometry of all models followed the described in section 2.2 for the proposed technology in the case study of 5-storey 2-span frame. The modelled structure has the same arrangement as the test performed in the University of Trento in order to profit from the same equipment for the experimental campaign.

ASSESSING THE STEEL LINK

The modelling procedure followed the one described in section 3.2. However, it required the use of more **complex 3D solids** since the plates had to be milled to accommodate the countersunk heads of the screws in 45º. Because this geometry, the plates were "grooved" around the drills in a rectangular shape in order to detach the inner part from the outer in order to be meshed separately (see figure 29). The elements were then tied together solving problems with compatibility between the different meshes and reducing the integration time since this strategy enabled the definition of a coarse regular mesh out of the region of the holes and the application of hex elements for meshing the hole zone. The inclination of the screws also influenced the **mesh** of beam, which had to account for more number of partitions to make the structure more regular.

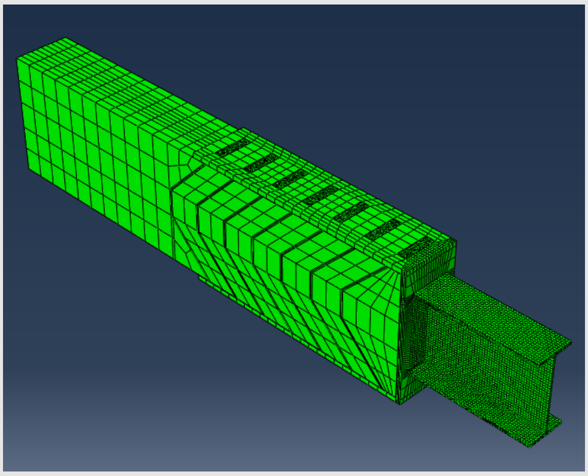


figure 29 – Steel link and proposed connection modelled with timber as isotropic

The input parameters were the same as the ones resulting from the calibration of the model for Trento's link for the **material mechanical properties** of S235. This specification was applied to all the plates and steel profile in the model. The material of screws in carbon steel was considered with the proper characteristic yield strength of 1000 MPa, also transformed to the true stress-strain curve. The timber elements were considered as isotropic, profiting from the same solution used for the calibrated test. In this way, it is assumed that under the specified loading conditions, the timber will behave elastically based of its longitudinal properties.

For the interaction between elements, the general contact was assumed as previously. Specially to define the contact between the screws and the embedment zone in timber, a tying constrain was defined considering there is no significant load-slip behavior of the connection, due to the design concept and to the application of full threaded screws in 45º. Tie for constraining movement of screws to the timber beam since there should be no significant slip of the connection to the beam.

The model was also developed in a cantilever configuration by fixing the link extremity and loading the free end of the timber beam. The controlled displacement produced the equivalent action of a monotonic load applied smoothly and the results were displayed in a moment-rotation capacity curve based in a history output.

ASSESSING THE CONNECTION

The pursued goal when performing this analysis is to develop a way to assess the response of the connection to the same type of load in the structural configuration. To reproduce the real behavior of connections with metal fasteners, the elements responsible for the load bearing capacity, in this study case: plate, screws and timber, need to be considered in an **appropriate model** that can reproduce the material response.

Aiming to perform a **simple analysis** that can be easily reproduced, it was determined that all the material mechanical properties would be defined through the available tools in Abaqus package. In particular, the Hashin Damage model was chosen as the being the most appropriate to define the failure mechanism of an also simplified wooden material.

Hence, a simplification of the structure is required in order to define elements with a **plane stress formulation**. In a general way, edgewise connections in timber beam mobilize only one plane formed by the longitudinal and radial axis. This simplification showed through the previous models that there is no lost in the correspondence with the real behavior once the main plane of load and bearing capacity lies in the same discretized layer.

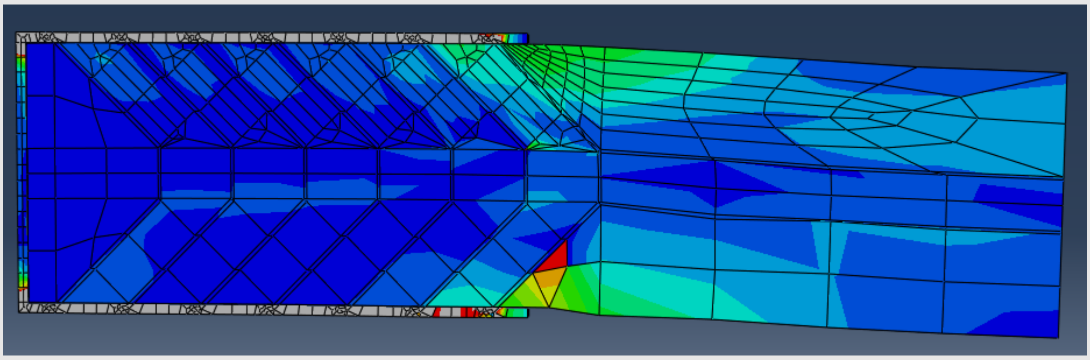


figure 30 – Connection model - Deformed shape amplified with a scale factor of 3

In this way, the modelling process followed the same procedure described in sections 3.4 and 3.5. However, the material properties were calibrated in the bending test performed with GL28c and the study case used for the design of the proposed technology was specified with GL28h. In this situation, it was necessary to consider the same **glulam** as the calibrated in bending test in order to ensure reliability in the result values of the prediction model. Since this material presents lower values of resistance, rigidity and density the predicted values are considered to be in a safe range.

Another consideration that should be made while dealing with calibrated values in different model configuration is the **mesh dependency**. In order to reduce the influence of mesh in the results the size of the elements were kept almost with the same size of seeds the number of shells and points of integrations were kept the same as the validated models form the dome connection as it can be observed in figure 30.

ASSESSING THE TIMBER BEAM

Fearing brittle failure to happen in the timber beam, a model was developed in the **gross cross-section** member of the timber in order to assess the available overstrength of the element in relation to the other component elements of the joint.

Since the modelling of timber in this thesis is focused in plan stress formulations, a model was performed on the same base of the connection model described here just before. Because of **mesh dependency** and material properties the same partitions, the same seeds, mesh controls and layers were used for this model, although with no screws modelled, the partitions followed the same as the previous model. This solution aims to give coherence between the values of the results.

Finally, the tested structural system consists of a glulam beam fixed on one edge and loaded on the other by a **controlled displacement** in which the load-displacement curve was registered. The obtained deformed shape can be seen in figure 31.

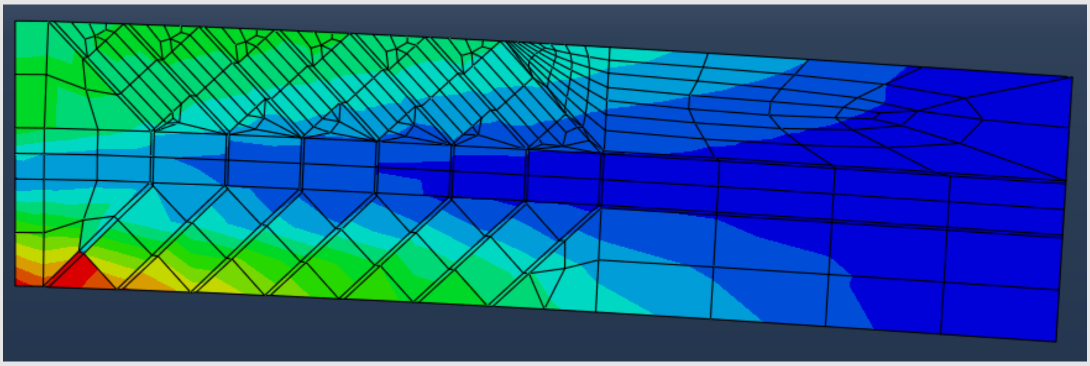


figure 31 – Glulam beam model - Deformed shape amplified with a scale factor of 3

From the results obtained from Abaqus, it can be observed that the failure of the beam starts on the top fibers of the glulam, where tensile strength is exceeded first. More details will be presented in the next section.

4.2. ANALYSIS OF RESULTS

The behavior of timber members in joint areas is influenced by the type of connection and by the development of a complex multiaxial stress state (Sandhaas, 2012). The use of inclined full threaded screws in connections enhances the capacity of the joint by causing friction between the connected elements and by triggering the withdrawal capacity of the screw. The study case used for the development of the models was designed based on the failure mode of the connection by a rotation of the fastener inside the timber element, developing 2 plastic hinges as it was mentioned in section 2.2. The establishment of a **screw failure** as the governing mode, is based in a combination of factors which involves the thickness of the plate, timber density, the diameter, the point side length, tensile strength and withdrawal capacity of the fastener. By observing the design conditions, it can be concluded that the steel plate is thick and the length of the screw in the timber member is long enough to activate the withdrawal resistance combined with the diameter and the tensile strength of the screw that are not enough to prevent deformations. Hence, the screw will develop plastic hinges and the deformation will occur around the middle length. The schema presented by Bejtka et al. showed in figure 32 shows the same failure mode of the inclined screw with the development of two plastic hinges, but in this case it represents the behaviour in timber-to-timber connections.

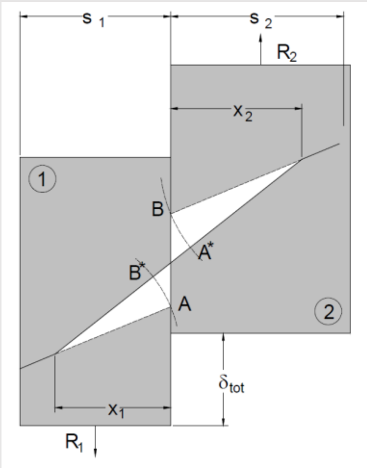


figure 32 – Displacements in a timber-to-timber connection with an inclined screw (Bejtka & Blass, 2002)

This complex ductile system was not the **governing failure mode** of the dome connection. A brittle failure in the timber beam was the result of the designed connection when subjected to a monotonic load. This leads to inconsistencies when just the dome connection was available to calibrate the failure parameters and to choose the adequate model. Since the test campaign based on the proposed configuration is not yet finished, the only experimental data used to calibrate the numerical model is the one of the dome connection.

By doing this, the analysis is once more considered in the safe side for the prediction of the behavior, once the model is calibrated on the brittle failure of timber. However, the **prediction model** will fail to provide an equivalent to the real rotation capacity of the connection because of the expected ductile failure mode. This question involves only the post-elastic behavior of the group. Hence, if it is verified that the overstrength is beyond the standard limits, it can be ensured that the connection and timber will remain in an elastic range, resulting in no difference for the moment-rotation capacity that is strictly dependent on the performance of the seismic steel link. With the results of the ongoing experimental campaign on the proposed technology new calibrations should be done in order to have a reliable model to optimize the solution.

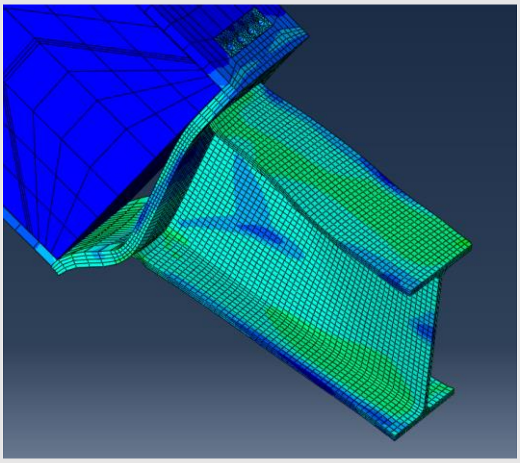


figure 33 – Proposed link - Deformed shape amplified with a scale factor of 3

Regarding all these considerations, the comparison of the analyses could be held and it can be concluded that the results of the numerical analysis seem to confirm the high potential of the proposed technology when subjected to monotonic loads. They show that the seismic link is the **weakest link** in the structural chain, being adequate with the capacity design concept and that it can develop a post-elastic behavior.

The first output to be analyzed is the failure mode obtained by the FEM, which final deformed shape is presented in the figure 33. Observing the response history of the steel link, the end-plate starts bending at the same time that tensile stresses occur at the bottom flange of the profile until it reaches its limit in tension strength. After this point, the top flange starts being stressed in compression and it can be seen a lateral instability in this post-elastic phase until it reaches a critical value of stresses. The final visual result is a totally deformed and distorted structure.

The curve **moment-rotation capacity** from this model presented in figure 34 shows clearly the two limits mentioned above. The first step down of the curve represents the loss of strength of the flange in tension. The gradual lost in lateral stability increases the moment resistance of the system and enhances the rotation capacity to high rotation values. The maximum values taken from this results are 192 kNm for bending moment and 0.17 rad for maximum rotation.

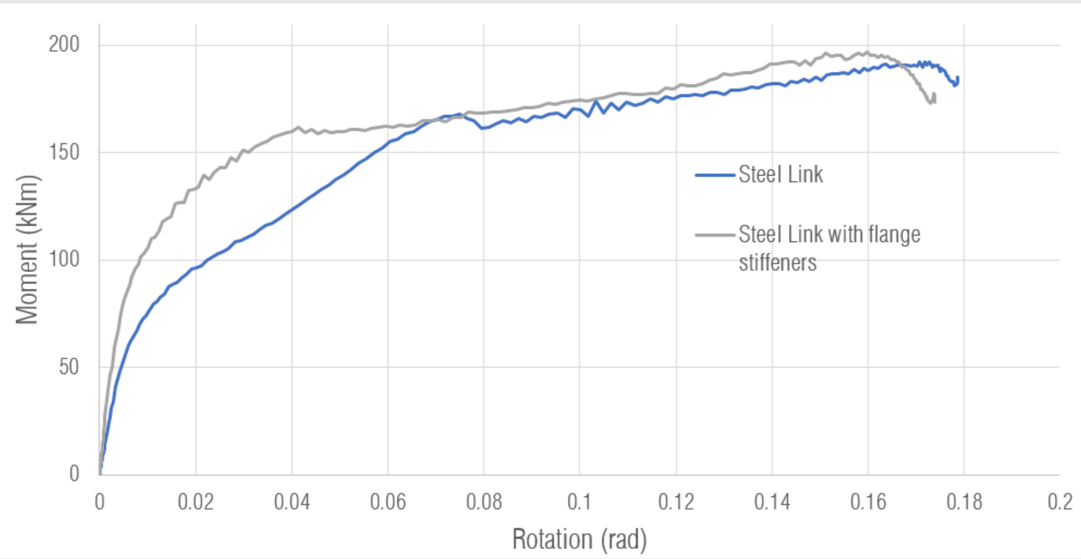


figure 34 – Moment-rotation curve of the proposed connection - Abaqus output

In a second stage, by analyzing the response of only the **proposed connection** subjected to a monotonic load, it could be verified that the failure mode is defined by a first rupture in tension leading to a complete failure due to compression stresses. Moreover, the maximum value obtained for the moment capacity of this composed system was 444 kNm, resulting in an overstrength of 2.31 to the maximum value extracted from the response of the structure with the seismic link.

Finally, the cantilevered **timber beam** was assessed and the response of the timber is a concentration of tensile stresses on the top lamellae that will conduct the element until brittle failure with a short achievement in displacement. The maximum moment obtained from this model was 273 kNm, 40% higher than the steel link.

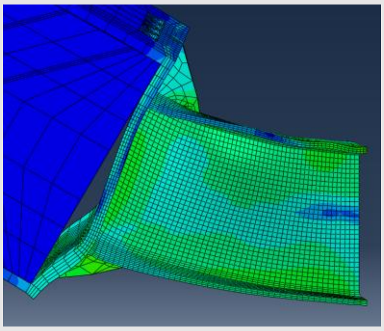


figure 35 – Proposed link with flange stiffeners - Deformed shape amplified with a scale factor of 3

It can be evaluated that the **ductile mechanism** will be favored in relation to the other elements involved in the connection, avoiding specially the brittle collapse, proof that they have sufficient strength. The energy is being dissipated by the steel link confirming the theoretical prediction. The results open a way to an improvement of this technology focused on the seismic resistant performance of timber structures.

By observing the results taken from this numerical analysis, it was possible to define some optimized solutions, like the one presented in figure 34 and figure 35 with the addition of flange stiffeners, on the top and on the bottom of the profile. This modification implied in a limitation of the rotation capacity but it enhances the moment capacity to a higher value and also increases the system stiffness.

5. CONCLUSIONS

This thesis was addressed to the development of a finite element model to predict the mechanical behavior of a steel profile inserted in a beam-to-column joint of a moment resistant timber frame. In case of a seismic event, the steel element was designed to act as the ductile element of the structure, dissipating energy through plastic deformations and keeping the connected elements in elastic range. A practical example of development this technology was presented in section 2.1 by considering a 5 storey-2 span building as a study case. The steel link, in this case, was connected to a glued-laminated beam by end-plate and edgewise plates with inclined full threaded screws as described in section 2.2. This solution presented a higher value of behavior factor than the one given as maximum in the Eurocode 8 for hyperestatic portal frames. Aiming to predict if the **seismic device** works as expected, the designed joint was modelled numerically.

In a first step to the modelling process, the validation models were developed to calibrate material values and failure modes based on monotonically experimental tests made for assessing a similar technology for the steel link and for the same type of connection. This stage requires the observation of the failure modes. When steel is the weak element in the structure, timber can be modelled as an isotropic material. However, when the timber governs the failure mode, its mechanical properties have to be defined for at least 2 directions, transforming wood into an equivalent transversally isotropic material.

The first set of validation models based on the yielding of plates and bars followed a straightforward process, in which there was enough information about the materials used for the production of the specimens. The unknown parameters were concentrated on the definition of the damage initiation and evolution. This data had to be calibrated on the experimental curves. Finally, the results obtained from Abaqus model showed a **good correlation** with the experimental results: for failure mode, stiffness, ultimate strain and moment capacity.

The models in which timber was responsible for the failure of the structure, special considerations needed to be done in order to transcript wooden mechanical behavior. This step of the study faced difficulties caused by the **lack of available data** on the materials used for the experimental tests and by the few accessible information on the definition of all the parameters necessary to develop a specific model to reproduce the tests results. The high number of variables that influences the final behavior of the structure implied a simplification of the material and of the analysis.

Once the seismic action is classified as an instantaneous type of loading, when defining material for the glulam, variations in the moisture and temperature were not taken into account. Other simplifications made in the system were consequence of the damage model selected to determine the failure envelope of the timber. "Hashin Damage" is applicable for plane-stress continuum elements and can define different values for compression and tension maximum stresses for longitudinal and transversal directions. Hence, the timber beam and plates were modelled in shells, following the plane defined by the row of screws, which represents the main orientation of stress distribution. The outer layers of the beam and plates, adjacent to the screwed slice, were tied to the former enabling the model to have the stiffness of the real specimen.

For the determination of an applicable model for assessing timber, FEMs were developed in order to qualify the modelling process as capable of developing an equivalent to the real structural behavior and approximate capacity of the tested structures. The methodology used was checked with the available experimental tests made on a similar connection for different types of loading: compression, tension and bending. The models could predict the failure mode and the maximum applied forces showing satisfactory results while the ultimate displacement was dependent of the post-elastic input for damage evolution, in special for the compression test. After the calibration on the tests results, all the models showed close approximation to the values in the tests. Once the methodology was validated, the calibrated parameters for the bending model were used to elaborate the predictive model for the proposed technology.

It should be mentioned that due to the high number of unknown data, the calibration procedure turned to be an extensive part of the modelling process. In addition to this, the use of a combined type of glulam for the beams produced a detachment of the material strength values from the standard values resulting in an increased number of variables for the calibration phase.

Because of the different approaches followed to determine the steel and timber behavior, the assessment of the link had to be done in parts. One oriented by the solid element-model in order to obtain a moment-rotation capacity curve and characterize the response of the steel link. Another model was focused on the evaluation of a maximum value for the moment capacity of the edgewise connection with inclined screws to the timber beam. This numerical analysis was developed with shell elements, as well as the third and last model performed to assess the response of the timber beam alone. The result from the group of predictive models confirms that the connection plate-screw-timber or only the timber beam have enough overstrength in relation to the moment capacity of the steel profile. This conclusion shows that the link is able to dissipate the seismic energy through inelastic deformations and prevent the brittle rupture in the timber elements.

The FEM of the connection is the starting point for the experimental campaign that is currently ongoing in the University of Trento. The future results will provide informations about the joint behavior and will also be used to calibrate the developed model, which will serve as a useful tool in the optimization of the solution.

Furthermore, since the models were calibrated with monotonic loading tests, in a second stage of the research it is expected the development of cyclic tests to complete the characterization of the mechanical behavior of the new technology.

6. ACKNOWLEDGMENTS

First of all, I would like to thank Prof. Dr. Eng. Federico M. Mazzolani, the mentor and supervisor of this research. I am very grateful for his support, critics and valuable orientation for the development of this thesis.

Second, I would like to thank the whole organization of SUSCOS Programme that gave me the opportunity to participate in it and for all the support they provided me.

Due to the technical support for modelling and refining the results in Abaqus, I am especially thankful to the assistant professor Mario D'Aniello. I would also like to thank the department colleagues for all the help received during the research.

I extend my gratitude to the eng. Zdeněk Dřevěný that contributed to this thesis by the several discussions about modelling processes.

Finally I would like to thank my family and friends that stood by me during all the period of this Master Course.

7. BIBLIOGRAPHY

Andreolli, M., Piazza, M., Tomasi, R., & Zandonini, R. (2011, April). Ductile moment-resistant steel-timber connections. *Structures and Buildings: Institution of Civil Engineers Proceedings*, pp. 65-78.

Bejtka, I., & Blass, H. J. (2002). Joints with inclined screws. *International Council for research and innovation in building and construction: Working commission W18 - Timber Structures.* Kyoto.

Branco, J. M. (2009). Portuguese traditional timber structures: survey, analysis and strengthening. *Protection of Historical Buildings* (pp. 261-266). Rome: CRC Press.

CNDB, C. N. (2011-2012). *Formation Bois Construction: Les obstacles au développement de l'utilisation du bois dans la construction et propositions d'actions.* Retrieved from Ministère de l'écologie, de l'énergie, du dévelopment durable et de la mer: http://www.developpement-durable.gouv.fr/IMG/pdf/DGALN_rapport_formation_bois_CNDB_311012.pdf

Crosatti, A., Piazza, M., & Tomasi, R. (2009). *Refurbishment of traditional timber floor with inclined screw connectors.* Rome: CRC Press/Balkema.

CWC - Canadian Wood Council. (2004). Energy and the Environment: in residential construction. *Sustainable Building Series Nº1*.

F&M-ingegneria. (n.d.). Retrieved January 11, 2014, from http://www.fm-ingegneria.com/project-detail.php?id=168

Fountain, H. (2013, September 23). *Towers of Steel? Look Again*. Retrieved from The New York Times: http://www.nytimes.com/2013/09/24/science/appeal-of-timber-high-rises-widens.html?_r=2&

Frangi, A. (2009, October 29). *Fire Resistance of Timber Structures.* Retrieved 01 11, 2014, from http://enem.dec.uc.pt/apresentacoes/Fire_Resistance_of_Timber_Structuras.pdf

Frangi, A. (2012, November 28). *Fire resistance assessment of timber structures.* Retrieved January 12, 2014, from http://eurocodes.jrc.ec.europa.eu/doc/2012_11_WS_fire/presentations/08-FRANGI-EC-FireDesign-WS.pdf

Gülkan, P., & Langenbach, R. (2004). The Earthquake Resistance of Traditional Timber and Masonry Dwellings in Turkey. *13th World Conference on Earthquake Engineering.* Vancouver.

Hong, J.-P. (2007). *THREE-DIMENSIONAL NONLINEAR FINITE ELEMENTMODEL FOR SINGLE AND MULTIPLE DOWEL-TYPEWOOD CONNECTIONS.* Columbia: THE UNIVERSITY OF BRITISH COLUMBIA.

Jaspart, J. P., Demonceau, J. F., Renkin, S., & Guillaume, M. (2009). *European Recommendations for the Design of Simple Joints in Steel Structures.* Mem Martins: ECCS.

Kuklík, P. (2008). Chapter 2 - Wood properties. In L. d. Project, *Handbook 1 - Timber Structures.* Educational Materials for Designing and Testing of Timber Structures - TEMTIS.

Laghdir, A., Fortin, Y., Cruz, C. M., & Hernández, R. E. (2008). Development of a technique to determine the 3d elasticity tensor of wood as applied to drying stress modeling. *Maderas. Ciencia y Tecnología: Universidad del Bío Bío, Chile - vol.10, núm. 1*, pp. 35-44.

Magrone, R. (2012, 02 10). *SOFIE project, seismic test 2007*. (CNR - IVALSA) Retrieved 01 24, 2014, from National Research Council of Italy: http://www.ivalsa.cnr.it/en/photo-video/foto/progetto-sofie-test-sismico-2007.html

Mazzolani, F. M., Faggiano, B., & Tartaglia, R. (2013). Ductile behavior of timber frames with steel links. *presentation in PowerPoint - COST FP1101 - Reinforcement of timber structures.* Trento.

Mazzolani, F. M., Faggiano, B., Tartaglia, R., & Krause, I. (2013). Ductile behavior of timber frames with steel links - new joint design. *presentation in PowerPoint - Rubner Holzbau.* Calitri.

Melzerová, L., Kuklik, P., & Šejnoha, M. (2012). Variable Local Moduli of Elasticity as Inputs to FEM-based Models of Beams made from Glued Laminated Timber. *32*, 425-434.

Metsä Wood. (2013, April). *Kerto Manual: Screwed Connections.* Retrieved from Metsä Wood: http://www.metsawood.com/materialarchive/MaterialArchive/Kerto-kasikirja-Ruuviliitokset_eng.pdf

Mirianon, F., Fortino, S., & Toratti, T. (2008). *A method to model wood by using ABAQUS finite element software: Part 1. Constitutive model and computational details.* Espoo: VTT Publications 687 - Technical Research Centre of Finland.

Piazza, M., Tomasi, R., & Modena, R. (2005). *Strutture in legno: Materiale, calcolo e progetto secondo le nuove normative europee.* Milano: Ulrico Hoepli Editore S.p.A.

République Française, M. d. (2010, June). *ETUDE COMPARATIVE SUR L'USAGE ET LE DEVELOPPEMENT DU BOIS.* Marne-la-Vallée: CSTB - Centre Scientifique et technique du batiment. Retrieved from

Ministère de l'écologie, du dévelopment durable et de l'énergie: http://www.developpement-durable.gouv.fr/IMG/pdf/DGALN_Etude_comparative_europeenne_bois_construction.pdf

Reynolds, T., Harris, R., & Chang, W.-S. (2012). DYNAMIC BEHAVIOUR OF TIMBER PERPENDICULAR TO THE GRAIN IN DOWEL-TYPE CONNECTIONS. *World Conference on Timber Engineering 2012 (WCTE 2012).* Auckland: Curran Associates, Inc.

Rosenfield, K. (2012, March 27). *Michael Green presents 'The Case for Tall Wood Buildings' 27MAR2012 by*. Retrieved from ArchDaily: Architecture News: http://www.archdaily.com/220779/michael-green-presents-the-case-for-tall-wood-buildings/

Ruggieri, N., Tampone, G., & Zinno, R. (2013). Typical Failures, Seismic Behavior and Safety of the "Bourbon system" with Timber Framing. *2nd International Conference on Structural Health Assessment of Timber Structures.* Trento: Trans Tech Publications Ltd.

Sandhaas, C. (2012). *Mechanical behaviour of timber joints with slotted-in steel plates.* Zutphen: Wöhrmann Print Service.

Sandhaas, C., & Ceccotti, A. (2012). *Earthquake resistance of multi-storey.* Retrieved from Forum Holzbau Beaune: http://www.forum-holzbau.ch/pdf/fbc12_sandhaas.pdf

Sandhaas, C., & Ceccotti, A. (2012). *Earthquake resistance of multi-storey massive timber buildings.* Retrieved from Forum Holzbau Beaune: http://www.forum-holzbau.ch/pdf/fbc12_sandhaas.pdf

Sandhaas, C., Kuilen, J.-W. v., & Blass, H. J. (2012). Constitutive Model for wood based Continuum Damage Mechanics. *World Conference on Timber Engineering.* Auckland.

Sandhaas, C., Vries, P. d., & Kuilen, J.-W. v. (2010). Double-shear timber joints with high strength steel dowels. *11th World Conference on Timber Engineering 2010 (WCTE 2010).* Trentino: Curran Associates, Inc.

Santos, C., Jesus, A. d., Morais, J., & Lousada, J. (2008). BEARING PROPERTIES OF PORTUGUESE PINE WOOD BENEATH A LATERALLY LOADED DOWEL. *Ciência e Tecnologia dos Materiais vol. 20 nº1/2*, pp. 45-51.

Skidmore, O. &. (2013). *Timber Tower Research Project.* Chicago: SOM.

Sousa, H. S., Branco, J. M., & Lourenço, P. B. (2013). *Glulam Mechanical Characterization.* Retrieved from Universidade do Minho: http://repositorium.sdum.uminho.pt/handle/1822/21217

SZS, C. s. (2011). Construction parasismique: Conception et dimensionnement. *03+04/11 steel doc*.

Techniker. (2014, 01 11). Retrieved 2014, from http://techniker.oi-dev.org/blog/view/tall-timber-buildings-the-stadthaus-hoxton-london

Thomson. (2014). Retrieved 01 11, 2014, from http://thomsonrealestate.com.au/2013/03/28/australias-tallest-wooden-apartments/

Tomasi, R., Zandonini, R., Piazza, M., & Andreolli, M. (2008). *Ductile End Connections for Glulam Beams.*

STANDARDS

EN 1194:1999. Timber structures - Glued laminated timber - Strength classes and determination of characteristic values. Brussels, CEN, 1999.

EN 1995 1-1:2004, Eurocode 5. Design of timber structures - Part 1-1: General - Common rules and rules for buildings. Brussels, CEN, 2004.

CEN Eurocode 8: Design of structures for earthquake resistance - Part 1: General rules, seismic actions and rules for buildings - Brussels, Belgium: European Committee for Standardization, 2004.

FINITE ELEMENT SOFTWARE PACKAGE

ABAQUS® Version 6.10, Dassault Systèmes Simulia Corp., Providence, RI, USA, 2010.