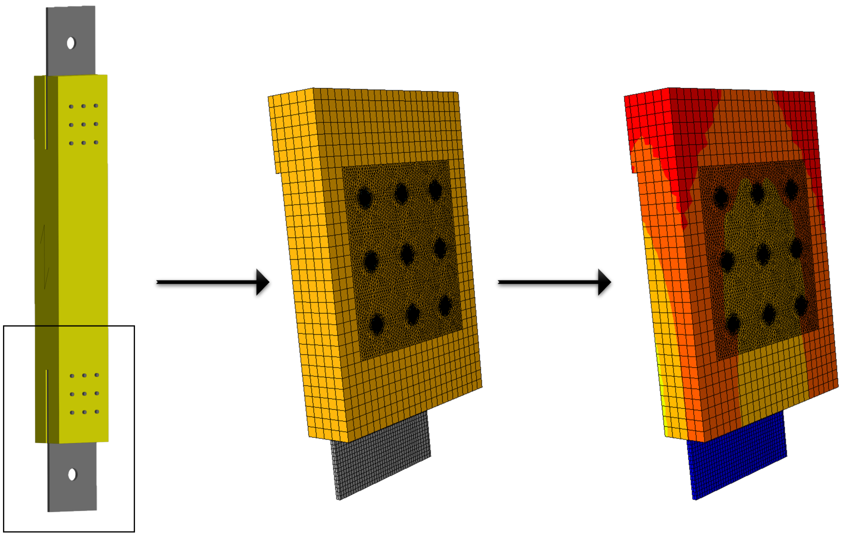
Master's thesis

NTNU
Norwegian University of Science and Technology
Faculty of Engineering
Department of Structural Engineering

Erik Hattestad
Trygve Vangsnes

An Experimental and Numerical Investigation of Stiffness in Dowel-type Connections

Master's thesis in Structural Engineering, TKT4950
Supervisor: Kjell Arne Malo
June 2022



NTNU
Norwegian University of
Science and Technology

Erik Hattestad
Trygve Vangsnes

An Experimental and Numerical Investigation of Stiffness in Dowel-type Connections

Master's thesis in Structural Engineering, TKT4950
Supervisor: Kjell Arne Malo
June 2022

Norwegian University of Science and Technology
Faculty of Engineering
Department of Structural Engineering

NTNU
Norwegian University of
Science and Technology

Department of Structural Engineering
Faculty of Engineering
NTNU – Norwegian University of Science and Technology

ACCESSIBILITY

CLOSED

MASTER THESIS 2022

SUBJECT AREA:	DATE:	NO. OF PAGES:
Timber structures	11.06.2022	75 + Appendices

TITLE:

An experimental and numerical investigation of stiffness in dowel-type connections

Eksperimentell og numerisk analyse av stivhet i dybelknutepunkt

BY:

Erik Hattestad
Trygve Vangsnes

SUMMARY:
Tall timber buildings will be an important part of the built environment in the future. Not only because the building material is environmentally friendly, but because building taller may be one of the best ways to deal with increasing urbanization. Timber structures are light weight structures sensitive to vibrations caused by dynamic loads. The stiffness of the connections in such structures is crucial in order to fulfill requirements regarding deflections and vibrations in serviceability limit state (SLS), such that the comfort level of occupants in the building is maintained.

One of the most used connection types in timber structures are dowel-type connections with slotted-in steel plates. The strength of such connections is thoroughly investigated and may be determined with a high degree of certainty. The stiffness, on the other hand, is associated with greater uncertainty. This makes it challenging to develop a numerical FE-model which gives a sufficient representation of the reality.

The scope of the present thesis was to produce experimental data as input to develop a fully parametric numerical FE-model. Three different dowel connections with different configuration, grain direction and strength class were exposed to cyclic tension load with varying load amplitude and mean load. The experimental data was firstly compared to theoretical values calculated with the Eurocode and secondly used to optimize the stiffness of two implemented rings close to the dowel in the numerical model. The rings were introduced in order to represent initial crushing and stiffness of a dowel-type connection.

The experimental results showed that the Eurocode for most cases underestimated the connection stiffness, i.e. the measured stiffness was higher than the theoretical. The stiffness increased for increased mean loading, and consequently gave bigger deviations between measured and theoretical stiffness. For the numerical model, the inner ring closest to the dowel proved to have larger influence than the outer ring. Numerical simulations without the reduced stiffness in the rings gave too stiff results. The numerical model from this study may be further developed and optimized, such that it can be used to determine the stiffness of a dowel-type connection of any geometry.

RESPONSIBLE TEACHER: Professor Kjell Arne Malo

SUPERVISOR(S): Professor Kjell Arne Malo

CARRIED OUT AT: Department of Structural Engineering, NTNU

Institutt for konstruksjonsteknikk
Fakultet for ingeniørvitenskap
NTNU – Norges teknisk-naturvitenskapelige universitet

TILGJENGELIGHET
LUKKET

MASTEROPPGAVE 2022

FAGOMRÅDE:	DATO:	ANTALL SIDER:
Trekonstruksjoner	11.06.2022	75 + Appendiks

TITTEL:

Eksperimentell og numerisk analyse av stivhet i dybelknutepunkt

An experimental and numerical investigation of stiffness in dowel-type connections

UTFØRT AV:

Erik Hattestad
Trygve Vangsnes

SAMMENDRAG:
Høye bygninger med treverk som bygningsmateriale vil være en viktig del av fremtidens konstruksjoner. Ikke bare fordi materialet er miljøvennlig, men fordi bygging i høyden er en av måtene å takle den stadig økende urbaniseringen på. Trekonstruksjoner er lette konstruksjoner som følgelig er sensitive for vibrasjoner som resultat av dynamiske laster. Stivheten til forbindelsene i slike konstruksjoner er avgjørende for å tilfredsstille krav til forskyvninger og vibrasjoner i bruksgrensetilstanden (SLS), slik at det ikke skal oppleves ubehagelig for mennesker som oppholder seg i konstruksjonene.

En av de mest brukte forbindelsene i trekonstruksjoner er dybelforbindelser med innslissede stålplater. Styrken til slike forbindelser er grundig utforsket, og kan fastsettes med høy grad av sikkerhet. Stivheten er det imidlertid knyttet større usikkerhet til, og dette gjør det utfordrende å lage en numerisk elementmodell som gir en tilfredsstillende representasjon av virkeligheten.

Hensikten med denne oppgaven var å gjennomføre eksperimentelle forsøk av dybelforbindelser som skulle gi data til utvikling av en fullparametrisk numerisk modell. Tre forskjellige dybelknutepunkt med ulik dybelkonfigurasjon, fiberretning og styrkeklasse ble utsatt for syklisk strekklast med varierende lastamplitude og middellast. De eksperimentelle resultatene ble først sammenlignet mot teoretiske verdier beregnet med Eurokoden og senere brukt for å optimalisere stivheten av to implementerte ringer nær dybelen i den numeriske modellen. Ringene ble introdusert for å representere initiell knusning og stivhet i en dybelforbindelse.

De eksperimentelle resultatene viste at Eurokoden i de fleste tilfeller underestimerte stivheten til forbindelsene, altså var den målte stivheten større enn den teoretiske. Stivheten til en forbindelse viste seg å bli større for store middellaster, og resulterte følgelig i større forskjell mellom målt og teoretisk stivhet. For den numeriske modellen viste ringen nærmest dybelen seg å ha større betydning enn den andre ringen. Numeriske simuleringer uten redusert stivhet i noen av ringene gav for stive resultater. Den numeriske modellen fra denne studien kan videreutvikles og optimaliseres slik at den kan brukes til å bestemme stivheten til et dybelknutepunkt av vilkårlig geometri.

FAGLÆRER: Professor Kjell Arne Malo

VEILEDER(E): Professor Kjell Arne Malo

UTFØRT VED: Institutt for Konstruksjonsteknikk, NTNU

Preface

The research presented in this thesis has been produced as part of the Master of Science in Civil and Environmental Engineering at The Norwegian University of Science and Technology (NTNU) for the two contributing authors. The work has been conducted during the spring semester of 2022 in Trondheim, within the Timber Structures Group, as part of the Department of Structural Engineering. The thesis corresponds to 30 credit points per person.

The extensive interest of timber structures was the reason this thesis came to be. Both authors wanted to conduct research that may be used in order to improve the knowledge within connections in timber structures. A desire to combine a practical and theoretical approach made a combination of experimental work and numerical modelling, which this thesis is a result of, perfect.

We would like to thank our supervisor, Kjell Arne Malo, for valuable guidance, discussions, theoretical background material and inspiration to choose this project. Also many thanks to all the workers in the laboratory at NTNU; without you the experimental work would not have been possible. Gøran Loraas and Terje Petersen helped us with preparing and producing all necessary parts, while Trond Auestad, Marius Østnor Døllner, Johan Fagervold and Thomas Uhlving made us able to conduct the experimental work. All of you made it inspiring, interesting and joyful to spend hours in the laboratory. Fellow students Olav Guddal, Stian Gundersen Raniszewski, Andreas Grøndahl Nourouzi and Anders Kastet all deserve an honourable mention for sharing useful knowledge and participation in professional discussions. Lastly, we would like to thank each other for good cooperation, not only through this master thesis, but thorugh five memorable years at NTNU.

Trondheim, 11.06.2022

Erik Hattestad

Trygve Vangsnes

Abstract

Tall timber buildings will be an important part of the built environment in the future. Not only because the building material is environmentally friendly, but because building taller may be one of the best ways to deal with increasing urbanization. Timber structures are light weight structures sensitive to vibrations caused by dynamic loads. The stiffness of the connections in such structures is crucial in order to fulfill requirements regarding deflections and vibrations in serviceability limit state (SLS), such that the comfort level of occupants in the building is maintained.

One of the most used connection types in timber structures are dowel-type connections with slotted-in steel plates. The strength of such connections is thoroughly investigated and may be determined with a high degree of certainty. The stiffness, on the other hand, is associated with greater uncertainty. This makes it challenging to develop a numerical FE-model which gives a sufficient representation of the reality.

The scope of the present thesis was to produce experimental data as input to develop a fully parametric numerical FE-model. Three different dowel connections with different configuration, grain direction and strength class were exposed to cyclic tension load with varying load amplitude and mean load. The experimental data was firstly compared to theoretical values calculated with the Eurocode and secondly used to optimize the stiffness of two implemented rings close to the dowel in the numerical model. The rings were introduced in order to represent initial crushing and stiffness of a dowel-type connection.

The experimental results showed that the Eurocode for most cases underestimated the connection stiffness, i.e. the measured stiffness was higher than the theoretical. The stiffness increased for increased mean loading, and consequently gave bigger deviations between measured and theoretical stiffness. For the numerical model, the inner ring closest to the dowel proved to have larger influence than the outer ring. Numerical simulations without the reduced stiffness in the rings gave too stiff results. The numerical model from this study may be further developed and optimized, such that it can be used to determine the stiffness of a dowel-type connection of any geometry.

Sammendrag

Høye bygninger med treverk som bygningsmateriale vil være en viktig del av fremtidens konstruksjoner. Ikke bare fordi materialet er miljøvennlig, men fordi bygging i høyden er en av måtene å takle den stadig økende urbaniseringen på. Trekonstruksjoner er lette konstruksjoner som følgelig er sensitive for vibrasjoner som resultat av dynamiske laster. Stivheten til forbindelsene i slike konstruksjoner er avgjørende for å tilfredsstille krav til forskyvninger og vibrasjoner i bruksgrensetilstanden (SLS), slik at det ikke skal oppleves ubehagelig for mennesker som oppholder seg i konstruksjonene.

En av de mest brukte forbindelsene i trekonstruksjoner er dybelforbindelser med innslissede stålplater. Styrken til slike forbindelser er grundig utforsket, og kan fastsettes med høy grad av sikkerhet. Stivheten er det imidlertid knyttet større usikkerhet til, og dette gjør det utfordrende å lage en numerisk elementmodell som gir en tilfredsstillende representasjon av virkeligheten.

Hensikten med denne oppgaven var å gjennomføre eksperimentelle forsøk av dybelforbindelser som skulle gi data til utvikling av en fullparametrisk numerisk modell. Tre forskjellige dybelknutepunkt med ulik dybelkonfigurasjon, fiberretning og styrkeklasse ble utsatt for syklisk strekklast med varierende lastamplitude og middellast. De eksperimentelle resultatene ble først sammenlignet mot teoretiske verdier beregnet med Eurokoden og senere brukt for å optimalisere stivheten av to implementerte ringer nær dybelen i den numeriske modellen. Ringene ble introdusert for å representere initiell knusning og stivhet i en dybelforbindelse.

De eksperimentelle resultatene viste at Eurokoden i de fleste tilfeller underestimerte stivheten til forbindelsene, altså var den målte stivheten større enn den teoretiske. Stivheten til en forbindelse viste seg å bli større for store middellaster, og resulterte følgelig i større forskjell mellom målt og teoretisk stivhet. For den numeriske modellen viste ringen nærmest dybelen seg å ha større betydning enn den andre ringen. Numeriske simuleringer uten redusert stivhet i noen av ringene gav for stive resultater. Den numeriske modellen fra denne studien kan videreutvikles og optimaliseres slik at den kan brukes til å bestemme stivheten til et dybelknutepunkt av vilkårlig geometri.

Contents

List of Tables

List of Figures

List of Abbreviations

NTNU	Norwegian University of Science and Technology
Glulam	Glued laminated timber
LVDT	Linear Variable Differential Transformer
CLT	Cross Laminated Timber
TTB	Tall Timber Buildings
DynaTTB	Dynamic Response of Tall Timber Buildings under Service Loads
SLS	Serviceability Limit State
FEM	Finite Element Method
EC5	Eurocode 5: Design of timber structures (CEN, 2004b)
CoV	Coefficient of Variation

List of Symbols

$\boldsymbol{a_1}$	Distance between fasteners in grain direction
$\boldsymbol{a_2}$	Distance between fasteners perpendicular to grain direction
$\boldsymbol{a_{3,c}}$	Distance from fastener to unloaded end
$\boldsymbol{a_{3,t}}$	Distance from fastener to loaded end
$\boldsymbol{a_{4,c}}$	Distance from fastener to unloaded edge
$\boldsymbol{a_{4,t}}$	Distance from fastener to loaded edge
$\boldsymbol{E_{mean}}$	Mean Young's modulus
$\boldsymbol{G_{mean}}$	Mean shear modulus
$\boldsymbol{F_{v,Rk}}$	Characteristic capacity per shear plane per fastener
$\boldsymbol{F_{90,Rk}}$	Characteristic capacity perpendicular to grain
$\boldsymbol{M_{y,Rk}}$	Characteristic yielding moment for fasteners
$\boldsymbol{f_{u,k}}$	Characteristic ultimate tensile strength of steel
$\boldsymbol{f_{h,0,k}}$	Characteristic embedment strength of wood in grain direction
$\boldsymbol{f_{m,k}}$	Characteristic strength based on bending moment in timber
$\boldsymbol{f_{t,0,k}}$	Characteristic strength in grain direction based on tension in timber
$\boldsymbol{t_1}$	Thickness of side member in contact with fastener
$\boldsymbol{b}$	Width of specimen
$\boldsymbol{d}$	Dowel diameter
$\boldsymbol{n}$	Number of fasteners per row parallel to grain
$\boldsymbol{n_{ef}}$	Effective number of fasteners per row parallel to grain
$\boldsymbol{A_{net}}$	Net cross-section area
$\boldsymbol{A_{net,t}}$	Net cross-section area perpendicular to grain
$\boldsymbol{A_{net,v}}$	Net shear area in grain direction
$\boldsymbol{K_{ser}}$	Serviceability limit state stiffness per shear plane per fastener
$\boldsymbol{\rho_{mean}}$	Mean density
$\boldsymbol{\rho_k}$	Characteristic density
$\boldsymbol{\epsilon}$	Strain
$\boldsymbol{\epsilon_e}$	Elastic strain
$\boldsymbol{\epsilon_p}$	Plastic strain
$\boldsymbol{\epsilon_i}$	Initial strain
$\boldsymbol{\sigma}$	Stress
$\boldsymbol{\sigma_u}$	Ultimate stress
$\boldsymbol{F_{Ed}}$	Applied load
$\boldsymbol{F_{Est}}$	Estimated capacity according to Eurocode 5
$\boldsymbol{F_a}$	Force amplitude
$\boldsymbol{F_{mean}}$	Mean load
$\boldsymbol{F_{max}}$	Maximum load
$\boldsymbol{F_{min}}$	Minimum load

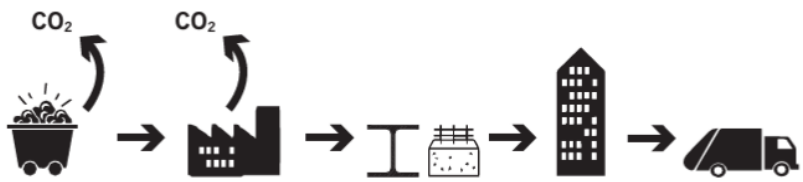
$\boldsymbol{\omega}$	Load frequency
$\boldsymbol{E_d}$	Dissipated energy
$\boldsymbol{\xi}$	Equivalent viscous damping
$\boldsymbol{Ds_{ue}}$	Strain based ductility
$\boldsymbol{\Delta u}$	Maximum displacement at failure
$\boldsymbol{F_{failure}}$	Maximum force at failure
$\boldsymbol{E_{r1}}$	Young's modulus in ring 1 in numerical model
$\boldsymbol{E_{r2}}$	Young's modulus in ring 2 in numerical model

Chapter 1

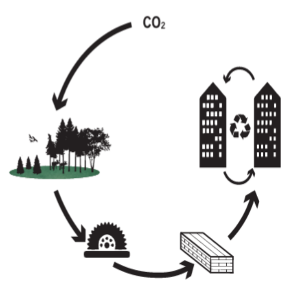
Introduction

From 1885 to 1931, the development of steel-framed tall buildings went from 10 to 102 stories (Foster et al., 2016). The same development, corresponding to an increase in number of stories bigger than 10 times, has not yet been found for timber buildings. However, only 11 years went by from the first wood-constructed nine-story building was completed, until Mjøstårnet, currently the world's tallest timber building, was completed with 18 stories and a total height of 85.6m (Abrahamsen, 2018). This has been possible due to evolution of engineering wood products, allowing longer spans and taller buildings (Green and Taggart, 2020). Exactly how tall timber buildings may be built in the future is hard to tell, but the fact that timber buildings will be important as residential buildings seems to be clear. Building in the height might be one of the best solutions to deal with urbanization and population expansion, due to the limited available space in urban areas.

Another reason for timber structures to be important in the future is to reduce carbon emissions from the building sector to reach the UNs climate goals (UNEP, 2020). While building materials such as steel and concrete follows a linear path of non-renewable materials, illustrated in figure 1.1a, engineering wood products is part of a renewable cycle consuming carbon dioxide, figure 1.1b. As sustainability-requirements is set to be even stricter in the upcoming years, usage of environmental-friendly materials will play a central role in the construction sector.



(a) Non-renewable material path (Green and Taggart, 2020)



(b) Renewable material path (Green and Taggart, 2020)

Figure 1.1: Life cycle of different building materials

The main concerns when building tall timber buildings (from now denoted TTB), are linked to wood's performance when exposed to fire and dynamic loads, such as wind or machinery. More precisely, the concern is with the connections joining the wooden parts together. To ensure reliable structures, not only the strength of connections, i.e. the load-carrying capacity, needs to be verified. The load-deformation behavior, i.e. the stiffness and ductility, is also an important parameter in earthquake situation as well as in everyday-usage, to ensure that vibrations and deflections does not exceed comfortable levels for the occupants (Sandhaas et al., 2020). In order to keep sway and acceleration of the building at an acceptable level, a common technique in TTBs is to create hybrid structures, combining either different engineering wood products and/or different materials, such as concrete. This method is utilized in the structural system of Mjøstårnet, including elements of glue laminated timber, cross laminated timber (CLT) and concrete slabs, see figure 1.2 (Abrahamsen, 2018). The truss along the height of the building is made up by glulam members joined together with dowelled connections, while the shaft is made of CLT. As timber structures are relatively lightweight structures with moderate stiffness and light damping, a small change in the structure's mass, stiffness or damping ratio may have a significant influence to the vibration response. This is the reason why the Trä8 lightweight floors were replaced by concrete slabs in the upper floors of Mjøstårnet.

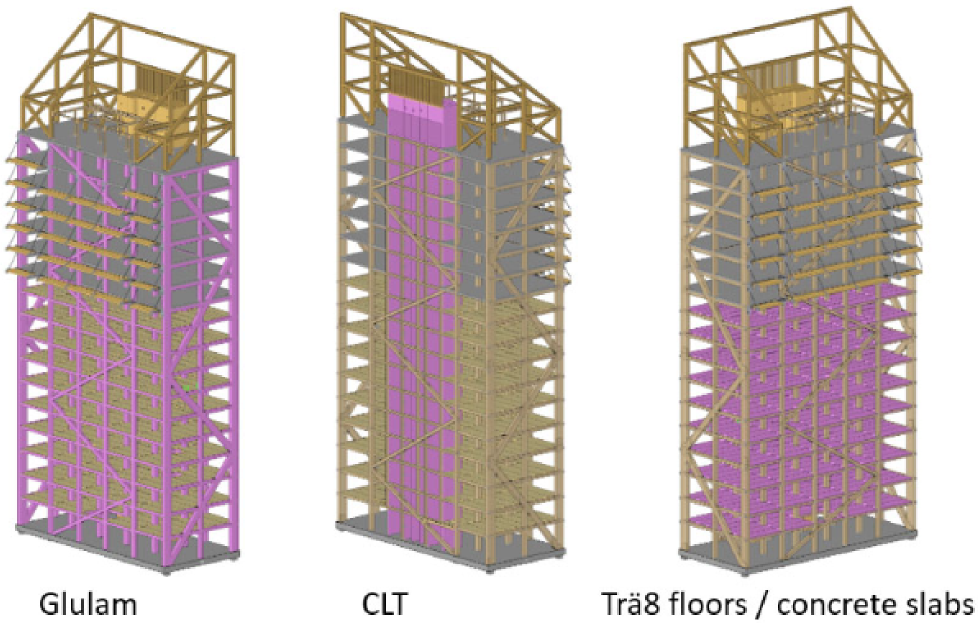


Figure 1.2: Stuctural system in Mjøstårnet (Abrahamsen, 2018)

.

Dowel-type connections are widely used in timber structures, and adds a certain stiffness to the structure. The difference between the timber parts and the connections is, however, the possibility to precisely determine the stiffness and damping contribution. This is due to the nonlinear behaviour of timber connections, even within the elastic range (Reynolds et al., 2014). This means that the same connection stiffness will not be displayed if a load is applied, removed and then reapplied. As the connection stiffness is dependant on the nature of the applied load, it is challenging for designers to model timber connections. Common practice is to replace the connections with equivalent linear stiffnesses, where the main challenge is to choose an appropriate stiffness making the structure neither too stiff nor too soft.

1.1 Motivation

The poor understanding and lack of modelling data of the response to timber structures exposed to dynamic loading, was the reason ForestValue started to fund the research program "DynaTTB - Dynamic Response of Tall Timber Buildings under Service Loads" (Abrahamsen et al., 2020). The project is driven by partners from research institutes, academia and designers in civil engineering companies from five different European countries. A total of five Work Packages (WP) make up the project, as illustrated in figure 1.3.

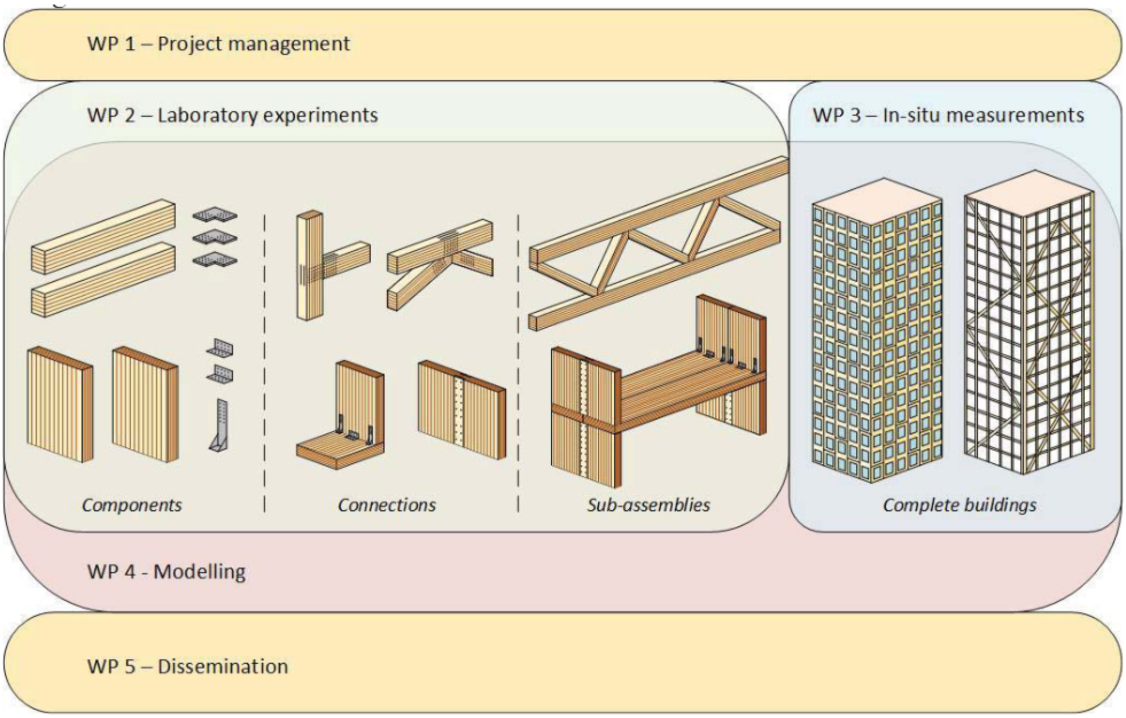


Figure 1.3: Work Packages in the DynaTTB project

WP2 and 3 are based on experimental work and includes testing of small-scale connections and components as well as measurements on existing TTBs, like Mjøstårnet. Full-scale testing is performed using electro-dynamic sliding shakers from the University of Exeter (UK) and CSTB (France), in order to create vibrations in the structures as a basis for establishment of Frequency Response Functions (FRF).

WP4 deals with FE-modelling of timber connections based on the data obtained from WP2 and 3. The overall objective is to develop a representative FE-model, such that vibration response of TTBs exposed to wind-induced dynamic loading may be predicted with an acceptable accuracy. One of the main uncertainties such models deals with, is where the stiffness and damping occur. It is clear that connections contribute to both stiffness and energy absorption, but there is yet little knowledge of their overall impact to TTBs. (Abrahamsen et al., 2020).

1.2 Background

The background for the present thesis was the master thesis of Frette et al. (2021), who performed experimental testing of small- and large scale dowel-type connections with slotted-in steel plates (see also section 2.7). In the present thesis only the small scale specimens were utilized, as the large scale specimens were ran till failure in the previous thesis. Before failure tests were performed, the specimens were exposed to cyclic tensile loading in order to investigate stiffness and energy dissipation. The main objective was to gather data that may be used for modelling aspects.

In addition to the experimental work, a numerical, fully parametric FE-model of dowel-type connections was developed. The experimental data was used to tune the FE-model, such that the numerical and experimental stiffness corresponded. The FE-model was limited to the elastic range, meaning crack propagation and failure modes were not implemented. Due to this, the present thesis deals with both WP 2 and 4 according to the DynaTTB-project.

1.3 Structure

The report has four main parts. The first part identifies the relevant theory concerning dowel-type connections, anisotropy in wood and a brief literature study addressing the state of the art within experimental and numerical analysis of dowel connections. Chapter 3 presents the experimental and the numerical method used in the present thesis, and the corresponding results is systematically included and discussed in chapter 4. Sources of error, further work and final conclusions are drawn in the last chapter.

Chapter 2

Theory

This chapter includes a basis of design for dowel-type connections according to the rules in Eurocode 5, including both capacity and stiffness calculations. Additionally, the orthotrop material model of wood for the purpose of numerical modelling. Lastly, a brief overview of the current state of the art research within dowel-type connection is presented.

2.1 Glulam

Glued laminated timber, commonly shortened glulam, is the oldest engineering wood product. The first industrial patented use was done by the German carpenter Otto Hetzer in the beginning of the 1900s (Stamatopoulos, 2021d). Glulam consists of multiple sawn timber boards, normally of softwood, glued together with strong adhesives. Finger joints at the end of the boards increases the contact surface between boards and allows long spans up to 30 meter. Even longer spans would, however, be possible, but are limited by transportation constraints. Glulam allows bigger cross sections compared to solid timber, and due to more uniformly distributed defects, glulam are less variable, resulting in higher characteristic strength (Bell, 2017).

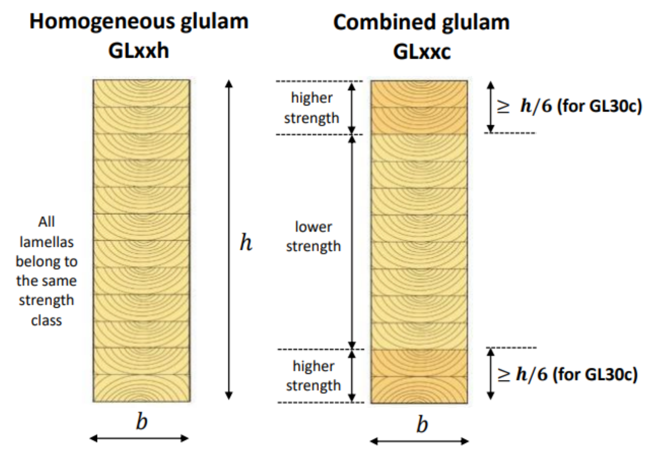


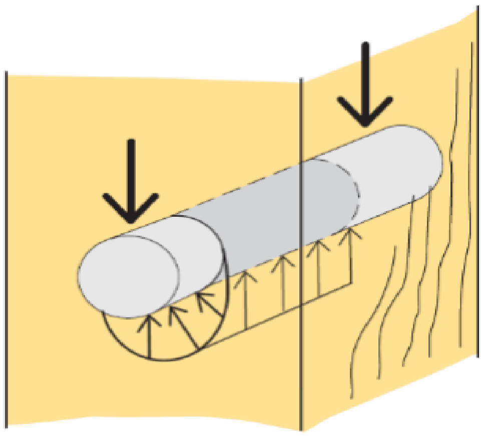
Figure 2.1: Overview of homogeneous and combined glulam (Stamatopoulos, 2021d)

There are two different types of glulam; homogeneous and combined, as illustrated in figure 2.1. Combined glulam consists of two types of lamellas, where the outer lamellas are of higher strength class than the inner, in order to efficiently carry stresses due to bending moments. In general, combined glulam has somewhat lower material properties than homogeneous glulam, where all the lamellas are made of the same strength class (CEN, 2013). Nevertheless, combined glulam is considered favorable in design in terms of efficient material usage. Notation for glulam is GLxxt, where xx denotes the characteristic bending strength and t represents glulam type by letter h and c for homogeneous and combined respectively.

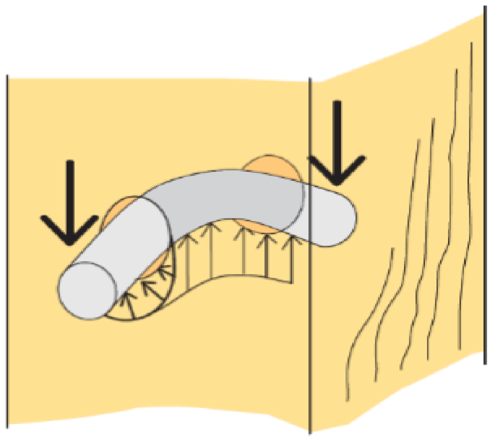
For GL30c, the outer lamellas are made of boards in strength class T22, while the inner lamellas are either made of T14 or T15. Timber in T-classes are classified based on the tensile strength and make up the basis for glulam (Stamatopoulos, 2021d).

2.2 Dowel-type connections

Dowels are smooth, solid cylinders, typically made of steel grades 4.6 to 10.9 used to connect timber elements in timber structures. Dowels are inserted through pre-drilled holes in the timber elements and may be applied in timber-to-timber or steel-to-timber connections (Bell, 2015). Dowels must have a diameter greater than 6 mm and smaller than 30 mm in order to fulfill the requirements to be considered as dowels in timber design. The most common diameter is 12 mm (Stamatopoulos, 2021a). As dowels are straight, cylindrical elements without a head, they can only be laterally loaded. The failure mechanisms for laterally loaded fasteners is either embedment of the sorrounding wood or yielding of the metal fastener. The first mecahnism typically applies for stocky fasteners, see figure 2.2a, while the second for slender fasteneres, figure 2.2b. (Stamatopoulos, 2021b)



(a) Embedment of wood



(b) Steel yielding

Figure 2.2: Failure modes in timber specimens (Stamatopoulos, 2021b)

Bell (2015) presents the following bullet points for efficient use of dowel-type connections:

- The center line of the connected components should meet in the same point, in order to avoid eccentricities.
- Quick and simple assembly
- Acceptable fire resistance
- Minimum amount of steel
- Standardized in a way which makes it possible to be used in several connections in a truss.

2.2.1 Slotted-in steel plates

Timber connections with slotted in steel plates are very suitable for large scale truss-constructions, with span up to 70-80 m (Bell, 2015). The possibility to add several steel plates gives rise to many shear planes, resulting in high capacity and stiffness per fastener (Bell, 2017). The steel plates may also be external, which in fact gives somewhat higher capacity, but not as favorable as internal steel plates in terms of fire design and aesthetics.

The most typical steel plate thickness is 8 mm, with slots normally being 2 mm greater (Bell, 2015). For one simple internal steel plate, the failure mode is not influenced by the thickness of the steel plate, while this is not the case for multiple slotted in steel plates (Stamatopoulos, 2021c). In those cases, the side and middle members are handled as single and double shear steel-to-timber members respectively, with either thin or thick external steel plates. Since the present thesis primarily deals with connections in double shear with internal steel plates, the guidelines in EC5-1-1 is presented for these connections only.

2.2.2 Minimum spacing

In order to achieve optimal strength and ensure ductile behavior of dowel-type connections, the minimum spacing requirements should be overheld. The minimum spacings are presented in table 2.1 according to EC5-1-1, §8.6, Table 8.5 (CEN, 2004b).

Table 2.1: Minimum spacings (CEN, 2004b).

Parameter	**Angle**	**Minimum value**
a_1 (parallel to grain)	$0 \leq \alpha \leq 360$	$(3 + 2\|\cos\alpha\|)d$
a_2 (perpendicular to grain)	$0 \leq \alpha \leq 360$	$3d$
$a_{3,t}$ (loaded end)	$-90 \leq \alpha \leq 90$	$\max(7d; 80\ \text{mm})$
$a_{3,c}$ (unloaded end)	$90 \leq \alpha \leq 150$	$a_{3t}\|\sin\alpha\|$
	$150 \leq \alpha \leq 210$	$\max(3,5\ \text{d}; 40\ \text{mm})$
	$210 \leq \alpha \leq 270$	$a_{3t}\|\sin\alpha\|$
$a_{4,t}$ (loaded edge)	$0 \leq \alpha \leq 180$	$\max((2 + 2\sin\alpha)d; 3d)$
$a_{4,c}$ (unloaded edge)	$180 \leq \alpha \leq 360$	$3d$

2.3 Capacity calculation: Eurocode 5

The load carrying capacity of laterally loaded fasteners is calculated by Johansen's equations and is covered in EC5-1-1, §8.2 (CEN, 2004b). These equations return the capacity per fastener per shear plane, denoted $F_{v,Rk}$. As the scope of the present thesis is limited to dowel-type connections with one simple slotted-in steel plate, the aspect of compatible failure modes are not presented here.

2.3.1 Failure modes

Three different failure modes are possible for connections with one internal steel plate. The brittle failure mode is represented by embedment of wood, while the ductile mode is represented by yielding of dowels with one or two plastic hinges (Geiser et al., 2021). The corresponding modes, denoted (f), (g) and (h) is illustrated in figure 2.3.

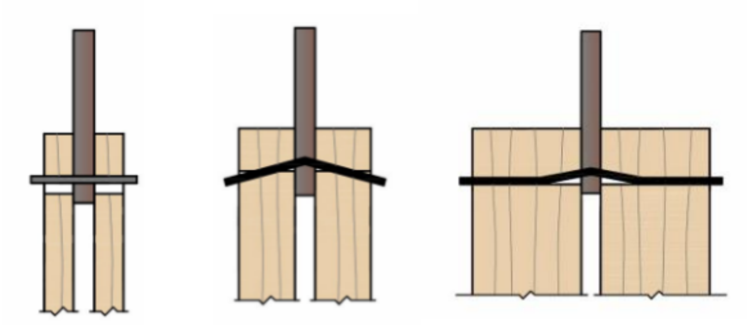


Figure 2.3: Failure modes for one internal steel plate (Stamatopoulos, 2021c)

The failure modes (f), (g) and (h) is presented in equation (2.1).

$$F_{\mathrm{v,Rk}} = \min \begin{cases} f_{\mathrm{h,k}} \cdot t_1 \cdot d & \text{(f)} \\ f_{\mathrm{h,k}} \cdot t_1 \cdot d \left[\sqrt{2 + \frac{4M_{\mathrm{y,Rk}}}{f_{\mathrm{h,k}} \cdot d \cdot t_1^2}} - 1 \right] + \frac{F_{\mathrm{ax,Rk}}}{4} & \text{(g)} \\ 2.3 \cdot \sqrt{M_{\mathrm{y,Rk}} \cdot f_{\mathrm{h,k}} \cdot d} + \frac{F_{\mathrm{ax,Rk}}}{4} & \text{(h)} \end{cases} \tag{2.1}$$

where t_1 is the thickness of the timber side member in contact with the dowels and d is the dowel diameter. The embedment strength of softwood, $f_{h,k}$, is given by:

$$f_{\mathrm{h,0,k}} = 0.082 \cdot (1 - 0.01 \cdot d) \cdot \rho_{\mathrm{k}} \tag{2.2}$$

where ρ_k is the characteristic density of timber. Equation (2.3) presents the yielding moment for dowels, $M_{y,Rk}$;

$$M_{\mathrm{y,Rk}} = 0.30 \cdot f_{\mathrm{u,k}} \cdot d^{2.6} \tag{2.3}$$

involving the ultimate tensile strength of the steel, $f_{u,k}$. In equation (2.1), the rope effect, $F_{ax,Rk}$, is contributing to increase the load-carrying capacity as it bends under laterally loading. The rope effect for dowel-type connections is, however, 0 %, as dowels can not carry axial loading, see section 2.2.

2.3.2 Splitting

In order to account for splitting parallel to grain, the load carrying capacity is multiplied by the number of effective fasteners in grain direction, n_{ef}.

$$n_{\mathrm{ef}} = \min \begin{cases} n \\ n^{0.90} \cdot \sqrt[4]{\frac{a_1}{13 \cdot d}} \end{cases} \tag{2.4}$$

Having that established, splitting parallel to grain will always be more critical than the load transfer as long as there are multiple fasteners in the grain direction.

2.3.3 Plug- and block shear

Another possible failure mode is plug- and block shear, described by the following set of equations in EC5-1-1, Appendix A (CEN, 2004b).

$$F_{bs,Rk} = \max \begin{cases} 1.5 A_{\mathrm{net},t} \cdot f_{t,0,k} \\ 0.7 A_{\mathrm{net},v} \cdot f_{v,k} \end{cases} \tag{2.5}$$

where

$$A_{net,v} = \begin{cases} L_{net,v} \cdot t_1 & \text{for failure modes (c, f, j/l, k, m)} \\ \frac{L_{net,v}}{2}\left(L_{net,t} + 2t_{ef}\right) & \text{for all other failure modes} \end{cases} \tag{2.6}$$

and

$$A_{net,t} = L_{net,t} \cdot t_1 \tag{2.7}$$

$L_{net,t}$ and $L_{net,v}$ is the total length in tension and shear respectively, according to figure 2.4. t_1 is the thickness of the wood or the contact length between dowel and wood if it does not correspond to the full thickness.

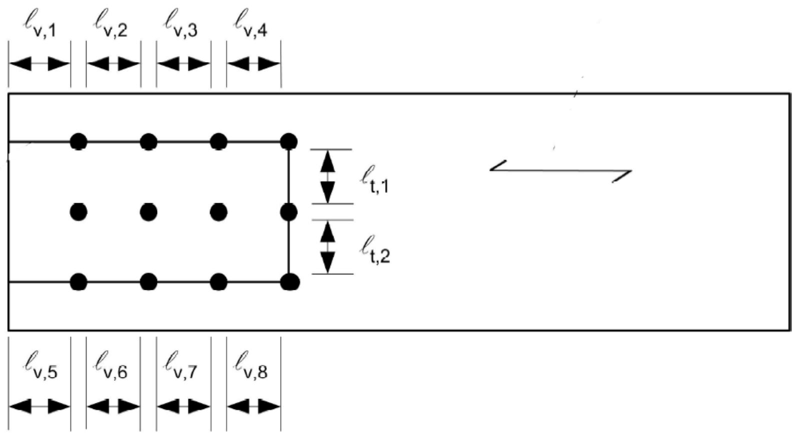


Figure 2.4: Relevant measures for calculation of block- and plugshear (CEN, 2004b)

The effective depth, t_{ef} is a parameter dependent on failure mode, given by:

$$t_{ef} = \begin{cases} 2\sqrt{\frac{M_{y,Rk}}{f_{h,k}d}} & \text{for failure mode (h)} \\ t_1\left[\sqrt{2 + \frac{M_{y,Rk}}{f_{h,k}dt_1^2}} - 1\right] & \text{for failure mode (g)} \end{cases} \tag{2.8}$$

2.3.4 Loading perpendicular to grain

When the load is applied perpendicular to grain, splitting may occur, see figure 2.5. The topic is covered in EC5-1-1, §8.1.4 and is calculated as in equation (2.9). The relevant geometry measures are taken from figure 2.5. $w = 1.0$ for all fasteners, except for punched metal fasteners (Stamatopoulos, 2021b).

$$F_{90,\mathrm{Rk}} = 14 \cdot b \cdot w \cdot \sqrt{\frac{h_{\mathrm{e}}}{1 - \frac{h_{\mathrm{e}}}{h}}} \tag{2.9}$$

There exists a more detailed way to calculate splitting perpendicular to grain, taking spacing between fasteners and multiple rows of fasteners into account, but for the present thesis, the capacity is limited to the rules presented in the Eurocode 5.

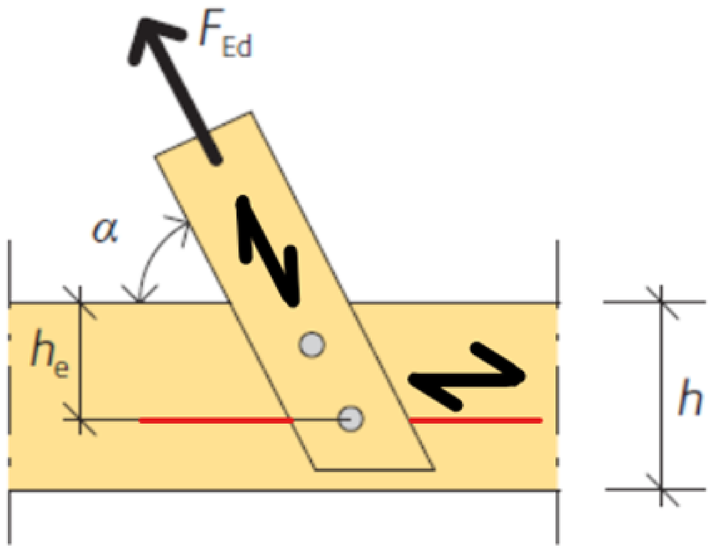


Figure 2.5: Splitting perpendicular to grain. Modified from Stamatopoulos (2021b)

2.4 Stiffness calculation: Eurocode 5

Slip modulus of connections with laterally loaded fasteners, i.e. dowels, is covered in EC5-1-1, §7.1, Table 7.1 (CEN, 2004b). Stiffness is, as load carrying capacity (see section 2.3), given in terms of per fastener per shear plane, in unit N/mm. Eurocode 5 gives the slip modulus for dowel-type connections under service loads as follows:

$$K_{\text{ser}} = \frac{\rho_m^{1.5} \cdot d}{23} \tag{2.10}$$

where ρ_m is the mean density and d the dowel diameter. To account for the fact that the steel contributes with additional stiffness, K_{ser} is multiplied with a factor of 2:

$$K_{\text{ser,steel-to-timber}} = 2 \cdot K_{\text{ser}} \tag{2.11}$$

The total stiffness of a component is modelled as a series of springs, see figure 2.6, resulting in a total stiffness given by equation (2.12).

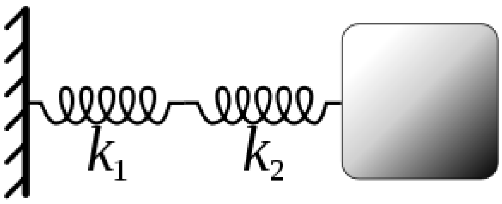


Figure 2.6: Spring in a series

$$K_{tot} = \left[\frac{1}{K_{ser,1}} + \frac{1}{K_{ser,2}}\right]^{-1} \tag{2.12}$$

2.5 Material model for wood

For elastic materials, the general form of Hooke's law in terms of stress-strain relationship reads as follows:

$$\sigma = \mathbf{C} \cdot \epsilon \tag{2.13}$$

where $\mathbf{C}$ is the stiffness matrix. When introducing a coordinate system where axis 1 represents the longitudinal direction (L) of wood, 2 the radial (R) and 3 the tangential direction (T) as shown in figure 2.7, equation (2.13) may be written as:

$$\begin{bmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{33} \\ \sigma_{23} \\ \sigma_{31} \\ \sigma_{12} \end{bmatrix} = \begin{bmatrix} C_{1111} & C_{1122} & C_{1133} & C_{1123} & C_{1131} & C_{1112} \\ C_{2211} & C_{2222} & C_{2233} & C_{2223} & C_{2231} & C_{2212} \\ C_{3311} & C_{3322} & C_{3333} & C_{3323} & C_{3331} & C_{3312} \\ C_{2311} & C_{2322} & C_{2333} & C_{2323} & C_{2331} & C_{2312} \\ C_{3111} & C_{3122} & C_{3133} & C_{3123} & C_{3131} & C_{3112} \\ C_{1211} & C_{1222} & C_{1233} & C_{1223} & C_{1231} & C_{1212} \end{bmatrix} \begin{bmatrix} \epsilon_{11} \\ \epsilon_{22} \\ \epsilon_{33} \\ \gamma_{23} \\ \gamma_{31} \\ \gamma_{12} \end{bmatrix}$$

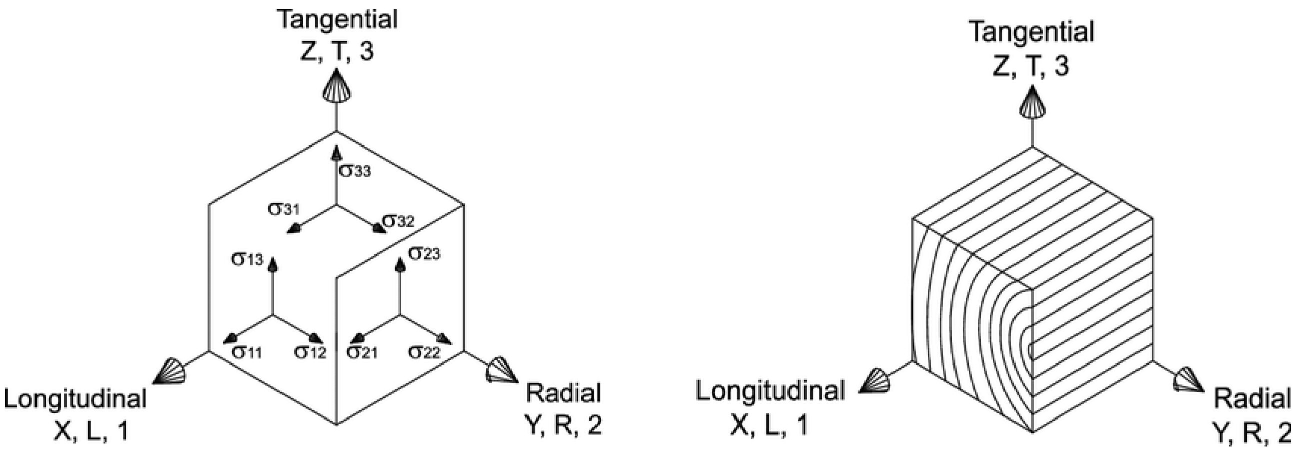


Figure 2.7: Stress components in wood (Carmen et al., 2020)

The stiffness terms C_{ijkl} is evaluated on the basis of the elastic potential, see equation (2.14)

$$C_{ijkl} = \frac{\partial^2 U}{\partial \epsilon_{ij} \partial \epsilon_{kl}} \tag{2.14}$$

Knowing that the order of differentiation do not influence the result, the stiffness matrix is symmetric due to $C_{ijkl} = C_{klij}$. In total, 21 unique coefficients is included in the general description of an anisotropic material.

2.5.1 Transformation of coordinate system

One coordinate system, x'_i, may be related to another coordinate system, x_j, through a transformation matrix a_{ij}, i.e. $x'_i = a_{ij} \cdot x_j$. The same transformation applies for strains, a second order tensor, and reads:

$$\epsilon'_{ij} = \epsilon_{pq} a_{pi} a_{gj} \tag{2.15}$$

As energy is an invariant property in terms of direction and does not change between which coordinate systems it is measured in, the stiffness terms between two coordinate systems can be expressed in the following way:

$$C'_{ijkl} = a_{qi}a_{rj}a_{sk}a_{tl}C_{qrst} \tag{2.16}$$

2.5.2 Orthotropy

A material with two or three mutually orthonormal planes of symmetry are called orthotropic. Such material is illustrated in figure 2.8.

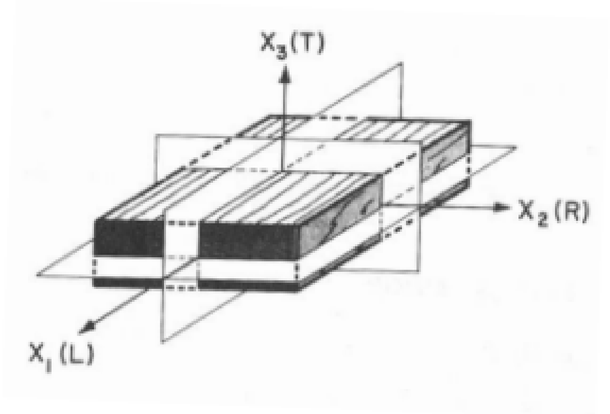


Figure 2.8: Orthotropic material with three planes of symmetry (Kjell Arne Malo, 2021a)

Assuming an orthotropic material with two symmetry planes in axis 1 and 2 gives rise to the following **a**-matrix.

$$\mathbf{a} = \begin{bmatrix} -1 & 0 & 0 \\ 0 & -1 & 0 \\ 0 & 0 & 1 \end{bmatrix}$$

By inserting values from the **a**-matrix above into equation (2.16) the stiffness terms in the **C**-matrix may be calculated for the symmetry plane.

$$C'_{1111} = a_{q1}a_{r1}a_{s1}a_{t1}C_{qrst} = (-1)\cdot(-1)\cdot(-1)\cdot(-1)\cdot C_{1111} = C_{1111} \tag{2.17}$$

$$C'_{1113} = a_{q1}a_{r1}a_{s1}a_{t3}C_{qrst} = (-1)\cdot(-1)\cdot(-1)\cdot(+1)\cdot C_{1113} = -C_{1113} \tag{2.18}$$

Equation (2.17) implies that nothing is changed, while equation (2.18) implies that the material property changes sign, which is impossible only due to symmetry. The same argument may be used for all stiffness terms with changed sign, resulting in the following expression of Hooke's law for orthotropic materials:

$$\begin{bmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{33} \\ \sigma_{23} \\ \sigma_{31} \\ \sigma_{12} \end{bmatrix} = \begin{bmatrix} C_{1111} & C_{1122} & C_{1133} & 0 & 0 & 0 \\ & C_{2222} & C_{2233} & 0 & 0 & 0 \\ & & C_{3333} & 0 & 0 & 0 \\ & & & C_{2323} & 0 & 0 \\ & \text{sym.} & & & C_{3131} & 0 \\ & & & & & C_{1212} \end{bmatrix} \begin{bmatrix} \epsilon_{11} \\ \epsilon_{22} \\ \epsilon_{33} \\ \gamma_{23} \\ \gamma_{31} \\ \gamma_{12} \end{bmatrix}$$

No coupling between shear and normal stresses resulting in the compliance matrix, **S** being of similar layout as the stiffness matrix. The compliance matrix relates strains in terms of stresses, i.e. $\epsilon = \mathbf{S}\cdot\sigma$. By applying uniaxial stress in each of the three directions separately, the compliance matrix may be established by introducing engineering constants, as Youngs modulus (E_{ij}), Poissons ration (ν_{ij}) and shear modulus (G_{ij}), where i and j represents plane and direction respectively. Finally, by inverting the compliance matrix, the stiffness matrix with engineering constant, is presented to be as follows (Kjell Arne Malo, 2021a):

$$C_{ij} = \begin{bmatrix} \frac{1-v_{23}v_{32}}{E_2E_3D} & \frac{v_{21}+v_{31}v_{23}}{E_2E_3D} & \frac{v_{31}+v_{21}v_{32}}{E_2E_3D} & 0 & 0 & 0 \\ & \frac{1-v_{13}v_{31}}{E_1E_3D} & \frac{v_{32}+v_{12}v_{31}}{E_1E_3D} & 0 & 0 & 0 \\ & & \frac{1-v_{12}v_{21}}{E_1E_2D} & 0 & 0 & 0 \\ & & & G_{23} & 0 & 0 \\ & sym. & & & G_{13} & 0 \\ & & & & & G_{12} \end{bmatrix},$$

where

$$D = \frac{1}{E_1E_2E_3}\begin{vmatrix} 1 & -v_{21} & -v_{31} \\ -v_{12} & 1 & -v_{32} \\ -v_{13} & -v_{23} & 1 \end{vmatrix} = \frac{1}{E_1E_2E_3}(1 - 2v_{21}v_{13}v_{32} - v_{23}v_{32} - v_{12}v_{21} - v_{13}v_{31})$$

2.6 Ductility

For solid materials that can be plastically deformed before reaching the fracture point, typically metals, the term ductility is associated with the material's elongation when exposed to uni-axial tensile loading (K. A. Malo et al., 2011). The same definition may, however, not be applied for timber engineering. In fact, there is no clear definition on how ductility shall be defined in timber design.

A typical stress-strain curve for a timber connection with metallic fasteners is shown in figure 2.9. The behavior of timber may be split into the following three parts:

1. A non-linear behaviour representing the initial strain slip, ϵ_i caused by clearance of the metallic fasteners. ϵ_i is found as the distance between zero and the interception between strain axis and the linearized part, see figure 2.9.

2. A linear elastic behavior according to Hookes law, see equation (2.13). ϵ_0 and σ_0 represents the yield strain and stress respectively, i.e. the point where the proportionality limit is reached.

3. A non-linear regime including the maximum loading point (σ_u, ϵ_u), fracture point (σ_f, ϵ_f) and maximum strain, ϵ_{max}.

Within the second phase, the dissipated energy during loading may be recovered during unloading. This is denoted elastic strain, ϵ_e, while the opposite, non-recoverable strain, is denoted plastic strain, ϵ_p. The total strain is a sum of the initial, elastic and plastic strain, see equation (2.19).

$$\epsilon = \epsilon_e + \epsilon_p + \epsilon_i \tag{2.19}$$

Rearranging equation (2.19) with respect to plastic strains gives $\epsilon_p = \epsilon - \epsilon_e - \epsilon_i$, which is a useful quantity when operating with ductility. An important measure is the permanent deformation at maximum load relative to the elastic deformation at the same load level, which leads to the following expression for strain based ductility (K. A. Malo et al., 2011).

$$Ds_{ue} = \frac{\epsilon_{pu}}{\epsilon_{eu}} = \frac{\epsilon_{pu}}{\sigma_u/E} \tag{2.20}$$

where ϵ_{pu} and σ_u represents the ultimate plastic strain and stress respectively. Ductility definitions based on energy may also be found in the literature, but this thesis will be limited to strain based ductility measures.

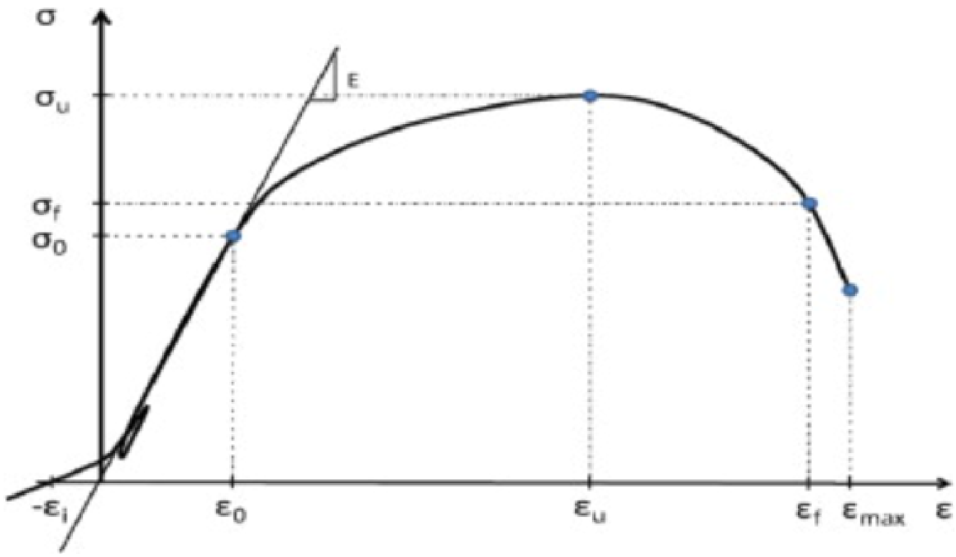


Figure 2.9: Generalized stress-strain curve for failure of timber connections (K. A. Malo et al., 2011)

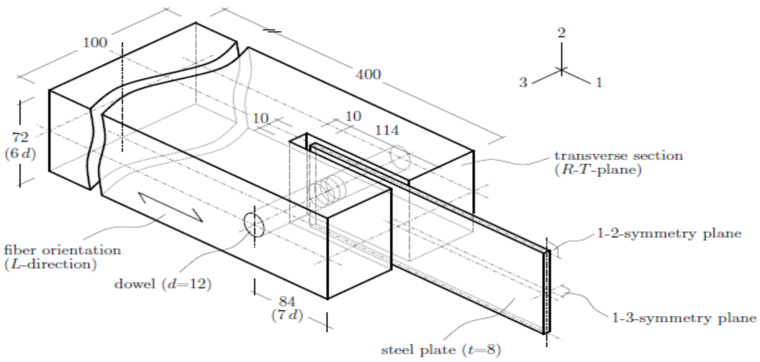
2.7 Literature review

Dorn (2012) combined experimental tests with numerical FE modelling of a single-dowel connection in order to investigate the stiffness. For the experimental part, a large number of parameters were varied, including end/edge distances, timber width, density and dowel roughness. An important aspect of the experimental study was to investigate the contact zone between wood and dowel. This was done by performing tests with both smooth and engrailed dowels. The results showed a high level of consistency where the roughened dowels increased the contact zone and the load-carrying capacity of the connections. For smooth dowels, the contact zone was estimated to be approximately half of the dowel diameter, resulting in high tensile stresses in the lateral direction due to a wedge-like action of the dowel. For rough dowels, the tensile stresses were reduced and replaced by increased shear stresses at a greater distance away from the symmetry plane. This resulted in a wood failure in the shear plane, tangential to the dowel hole. The change of contact zone resulted in a more ductile behavior due to the surrounding wood being crushed under compression in the shear plane, rather than being prone to brittle along-the-grain splitting under high tensile stresses.

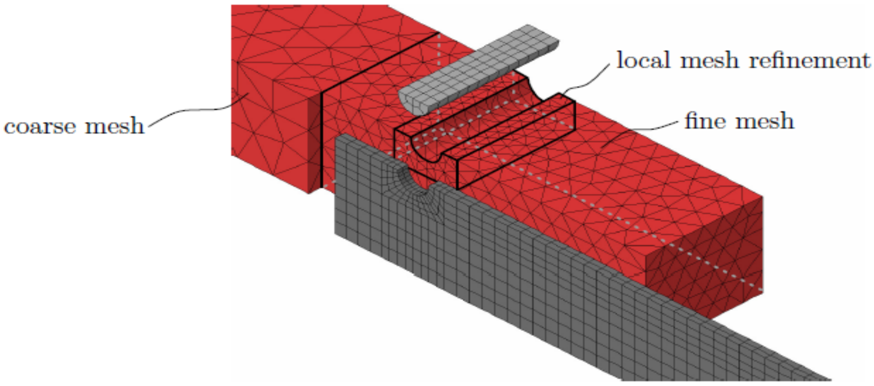
Another reason why the load-carrying capacity increased for rough dowels were due to a higher degree of bending, resulting in some kind of rope effect, which is stated to be zero for dowel-type connections (see section 2.3.1).

As load transfer in a dowel-type connection works through contact between wood and dowel, Dorn (2012) also investigated the influence of the surface roughness of the wood. This was carried out using five different cutting styles, involving circular saw with new and worn-out blades and CNC-machine. As for dowels, the results showed a higher compatibility for rougher surfaces, resulting in increasing deformations and consequently reduced stiffness.

For the numerical modelling, ABAQUS version 6.11, was used. The reference geometry, the basis for comparison for the later performed parametric study, is displayed in figure 2.10a. The corresponding mesh, with local refinements in the vicinity of the dowel, is shown in figure 2.10b. The element size varies from 3.5 to 14 mm for the timber part and 1.2 to 5 mm for the steel part, depending on distance away from the dowel.



(a) Reference geometry in FE-model



(b) Mesh in FE-model

Figure 2.10: Basis of FE-model (Dorn, 2012)

Dorn (2012) modelled the wooden part with transversal isotropic material behavior, and used the built-in contact modelling feature for the dowel-to-wood and dowel-to-steel plate part, with frictional coefficients of $\mu = 0.4$ and $\mu = 0.7$ respectively. The results of the reference model showed a maximum stiffness of 40kN/mm, reached at a load of 6.2kN. Furthermore, the results showed first yielding of dowel and wood at load of approximately 4.8kN and 5.5kN (Dorn, 2012). Tests investigating differences between orthotropic and transversal isotropic material model were carried out, resulting in minor differences.

The master thesis of Frette et al. (2021) investigated stiffness, K_{ser}, and viscous damping ratio, ξ, for three small-scale specimens of dowel-type connections exposed to service loads, see figure 2.11. The dowel configurations were varied throughout the experimental work. The specimens were exposed to pure tension and compression cyclic loads before a separate test series of fully reversed loading were carried out. The load magnitude was between 10 % (F_{min}) and 40 % (F_{max}) of the estimated capacities (F_{est}) for each specimen for the pure tension and compression loading. For each test, a number of 10 cycles were performed in order to achieve at least five stable cycles.

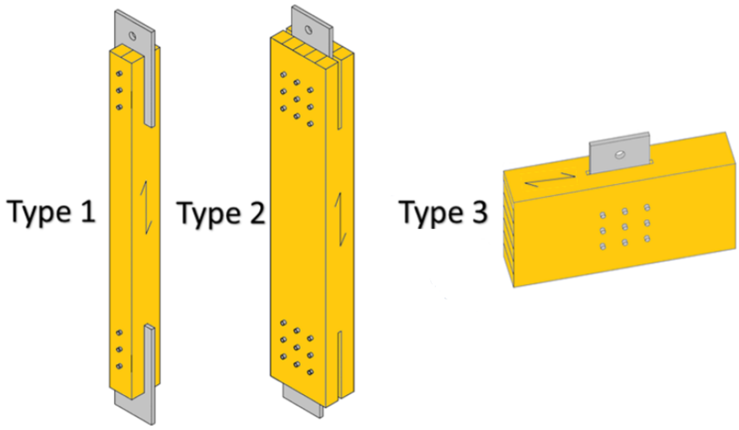


Figure 2.11: Small scale specimens used in the master thesis of Frette et al. (2021)

A summary of the results for full dowel configuration for each specimen under tension loading is presented in table 2.2. The stiffness values are given per shear plane per fastener, i.e. 2 shear planes and 3 and 9 dowels for specimen type 1 and type 2/3 respectively. The presented results is limited to the tension tests only, in order to present data that may be compared to the data obtained in the present thesis.

Table 2.2: Highlighted results from tension tests (Frette et al., 2021).

Specimen	$\boldsymbol{F}_{est}$ **[kN]**	$\boldsymbol{F}_{max}$ **[kN]**	$\boldsymbol{F}_{min}$ **[kN]**	$\boldsymbol{K}_{ser}$ **[kN/mm]**	$\boldsymbol{\xi}$ **[-]**
Type 1	49	19,6	4,9	17,42	0,073
Type 2	140	56	14	12,12	0,070
Type 3	78	31,2	7,8	6,02	0,029

Reynolds et al. (2014) applied cyclic service loads to single-dowel connections parallel and perpendicular to grain in order to describe vibration response for timber structures. Instead of measuring the stiffness of the connections only, the stiffness of the entire component was investigated, by measuring displacement from the fixed end of the specimen to the steel plate. The specimens were exposed to 1000 cycles of loading corresponding to 20 % and 40 % of their estimated capacities. The results showed a clear tendency of convergence of stiffness after 500-700 cycles (Reynolds et al., 2014). For the along-the-grain tests, an observation of increased stiffness for increased peak value was done, suggesting improved contact between dowel and wood. The same observation was, however, not done for tests perpendicular-to-grain, meaning that the plastic process in the contact zone appears faster than for parallel-to-grain.

If the ultimate load level is considered when determining the stiffness in the standard, the standard will return too conservative values for serviceability limit state. This is one of the main findings by Sandhaas et al. (2020), who collected state-of-the-art research and compared it to different standards, such as Eurocode 5. The reason for this was that dowel-type connections showed strong dependency to the respective load level.

Chapter 3

Method

This chapter is separated into two main parts, one addressing the experimental method and the other the numerical method.

3.1 Experimental method

This section provides descriptions about the performed experimental method. Specimens, material properties, calculated capacities and stiffnesses are presented in addition to the load procedure and post-processing of data.

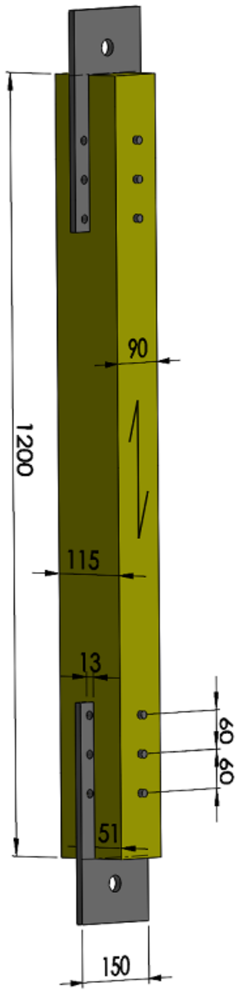
3.1.1 Prework

The specimens originate from the master thesis of Frette et al. (2021) which is briefly illustrated in figure 2.11. A detailed overview of the specimens is presented in appendix E. The prework mainly dealt with modifying the previous setup in order to run failure tests on the specimens. The steel plates used in specimen type 2 in the previous experimental work were replaced by new steel plates with increased capacity in order to withstand forces of 400 kN, which was the maximum force of the hydraulic actuator used in the cyclic and failure tests for specimen type 2. A larger cylindrical pin of Ø40 in the top and bottom connection was used to have sufficient shear capacity, and extra steel was welded on each side of the main steel plate to achieve necessary bearing capacity. Further calculation details of the steel plates may be seen in appendix F. For specimen type 3, the same steel plate as in Frette et al. (2021) was used because the specimens were tested in an actuator with a limited maximum force of 100 kN. All steel plates were prepared in the laboratory at NTNU and the respective dimensions are shown in figure 3.2.

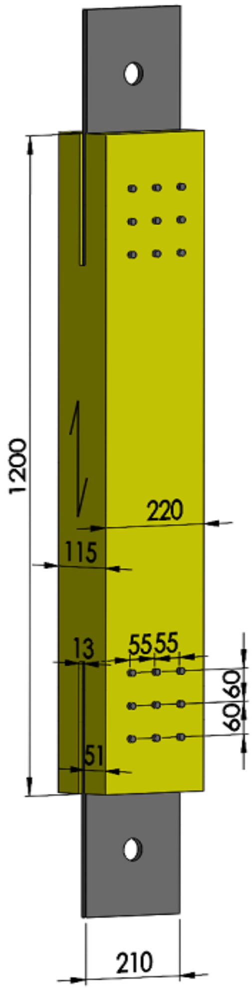
The capacities of the connections, F_{est} were calculated according to EC5 by theory presented in section 2.3. Fully detailed calculations may be seen in appendix F. The limiting capacities and corresponding SLS-stiffnesses, K_{ser}, is presented in table 3.1.

Table 3.1: Calculated capacities and stiffnesses for the specimens

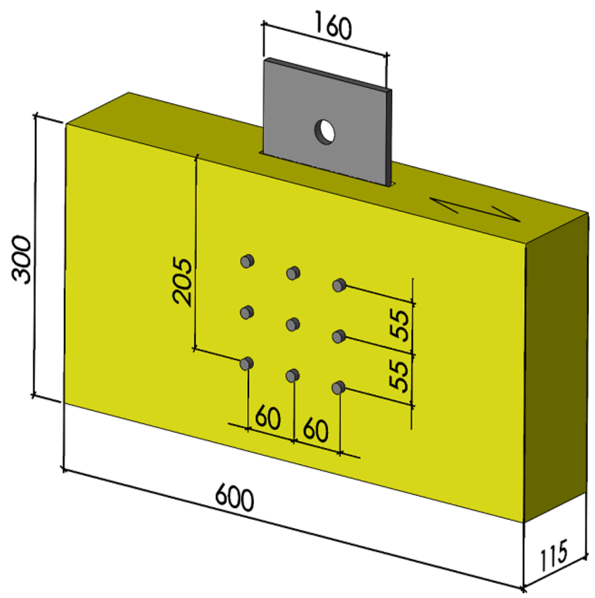
Specimen	$\boldsymbol{F_{est}}$ **[kN]**	$\boldsymbol{K_{ser}}$ **[kN/mm]**	**Predicted failure mode**
Type 1	55.6	63.8	Splitting ∥ grain, mode (g)
Type 2	157	167.5	Splitting ∥ grain, mode (g)
Type 3	72.7	167.5	Splitting ⊥ grain



(a) Specimen type 1

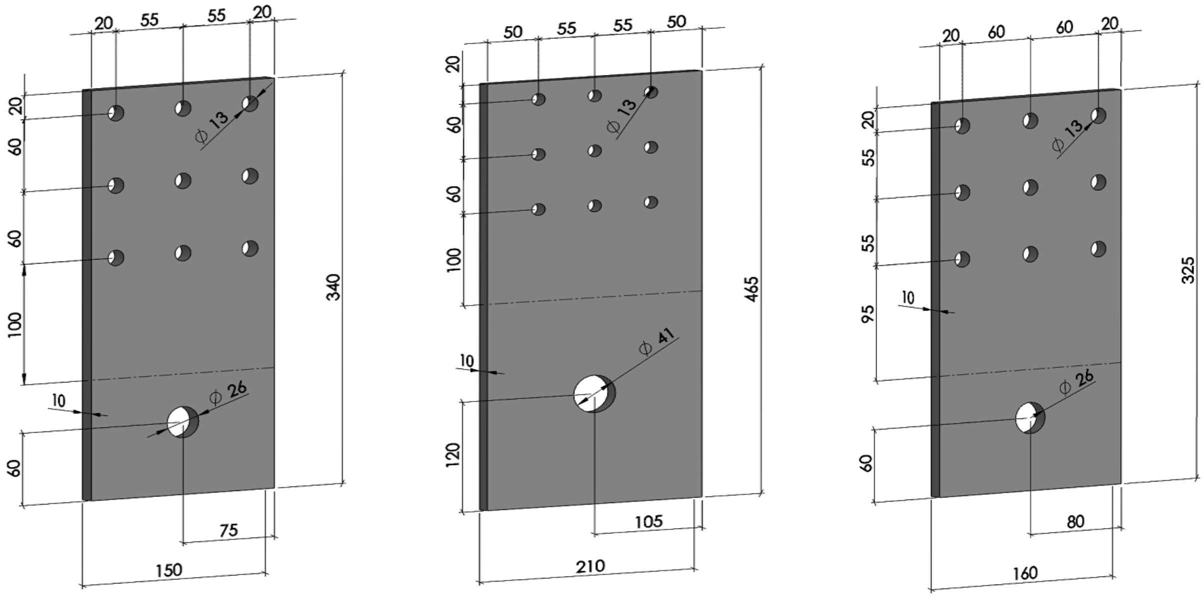


(b) Specimen type 2



(c) Specimen type 3

Figure 3.1: Specimens with dimensions used in experimental work



(a) Type 1 (b) Type 2 (c) Type 3

Figure 3.2: Steel plates used in specimens type 1, 2 and 3.

The specimens (type 1, 2 and 3) were cut from a GL30c-beam produced by Moelven Limtre consisting of inner lamellas of strength class T15 and outer of T22, all of thickness 45 mm (Frette et al., 2021). Specimen type 1 is sawn from the outer part (i.e. T22), while type 2 and 3 from the inner part (i.e. T15). Material properties for the timber parts are provided in table 3.2

Table 3.2: Timber properties according to NS-EN 14080 (CEN, 2013)

Material	$f_{m,k}$ $\left[\frac{N}{mm^2}\right]$	$f_{t,0,k}$ $\left[\frac{N}{mm^2}\right]$	G_{mean} $\left[\frac{N}{mm^2}\right]$	E_{mean} $\left[\frac{N}{mm^2}\right]$	ρ_k $\left[\frac{kg}{m^3}\right]$	ρ_{mean} $\left[\frac{kg}{m^3}\right]$
GL30c	30	19.5	650	13 000	390	430
T15	22	15	720	11 500	360	430
T22	30.5	22	810	13 000	390	470

The dowels were reused from Frette et al. (2021). For the failure tests, some new dowels had to be produced as some of them were bent during testing. These dowels were made of the same batch. Steel properties are presented in table 3.3.

Table 3.3: Timber properties according to NS-EN 14080 (CEN, 2013)

Object	d/t [mm]	l [mm]	$f_{y,k}$ $\left[\frac{N}{mm^2}\right]$	$f_{u,k}$ $\left[\frac{N}{mm^2}\right]$
Dowels	12	130	755	916
Steel plates	10	-	355	430-550
Cylindrical pin	26/40	-	355	430-550

3 samples of specimen type 1 and 3 and 4 samples of specimen type 2 were tested, each specimen denoted SX-Y, where X = 1,2,3 and Y = 1,2,3,4. The work flow of the experimental method is briefly illustrated in figure 3.3.

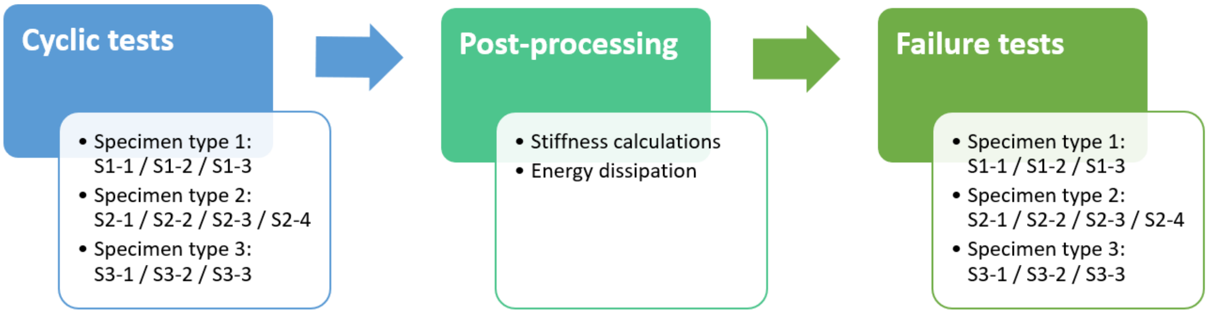


Figure 3.3: Work flow experimental method

3.1.2 Load procedure

The load procedure is modified from Kjell Arne Malo (2021b) and Frette et al. (2021). In order to test how the stiffness was influenced by different load levels according to the hypothesis of Sandhaas et al. (2020), a testing regime in tension was conducted. This limited the possibility to investigate slip in the connections, which requires a change in loading direction, i.e. fully reversed loading. Pure tension testing gives, however, a good representation of in-service loads such as wind, machinery and footfall. The wind load is simulated by adding a force in tension and the turbulence induced by the wind load is simulated through a cyclic sinusoidal load, centered around the mean applied tension load.

The load procedure is divided into four load phases. The load procedure for specimen type 1 is presented in figure 3.4, but looks, in principle, exactly the same for specimen type 2 and 3. All load procedures with corresponding calculations is included in appendix F. In order to achieve at least five stable cycles, taking into account that the first cycles might behave differently due to initial consolidation in the interface between dowel and wood (Dorn, 2012), 10 cycles were performed for each load phase. According to Kjell Arne Malo (2021b), the applied maximum load should not be greater than 40 % of the estimated capacity, F_{est}. The minimum load is, on the other hand, set to be 10 % of the estimated capacity, in order to obtain results that may be compared to results of Frette et al. (2021), see table 2.2 in section 2.7. The increasing load magnitude in each phase of the load procedure is developed to investigate the influence of load level and the corresponding stiffness, as presented in section 2.7. The load phases are as follows:

- Load phase 1: 10 to 20 % of F_{est}
- Load phase 2: 20 to 30 % of F_{est}
- Load phase 3: 30 to 40 % of F_{est}
- Load phase 4: 10 to 40 % of F_{est}

Table 3.4 presents the calculated load amplitudes (F_a), mean loads (F_{mean}) and load frequency (ω) for each specimens in each load phase.

The choice of load frequency of ω = 0.1 Hz was done on a basis of a suggestion by the machine operator in order to keep control of the machine and avoid machine limitations. The chosen load frequency is, however, below the preferred of 1 Hz, but still within the recommended range 0.1-2 Hz (Kjell Arne Malo, 2021b). The machine was operated in load control in all four phases.

Table 3.4: Load values for load procedure

Specimen	**Load phase**	F_{mean} [kN]	F_a [kN]	ω [Hz]
Type 1	1	8.34	2.78	0.1
	2	13.9	2.78	0.1
	3	19.46	2.78	0.1
	4	13.9	8.34	0.1
Type 2	1	23.55	7.85	0.1
	2	39.25	7.85	0.1
	3	54.95	7.85	0.1
	4	39.25	23.55	0.1
Type 3	1	11.7	3.9	0.1
	2	19.5	3.9	0.1
	3	27.3	3.9	0.1
	4	19.5	11.7	0.1

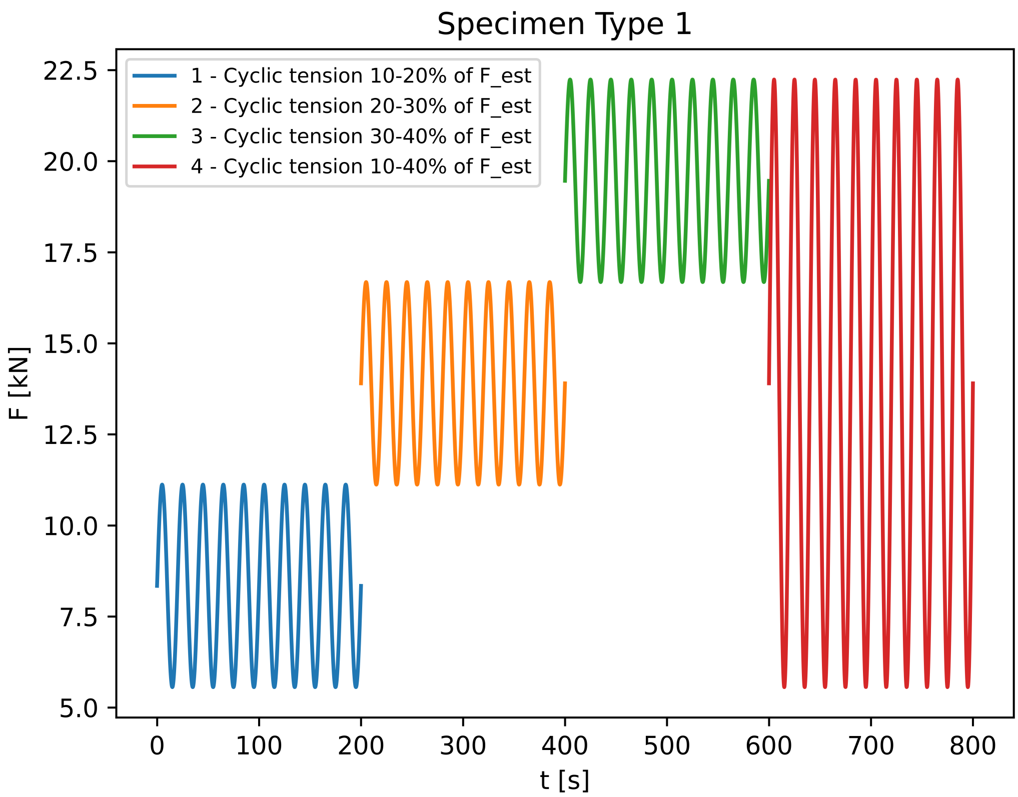


Figure 3.4: Load procedure specimen type 1

After the cyclic tests, a failure test was ran according to NS-EN 26891. First, the specimen was loaded under constant rate, $0.2 \cdot F_{est}$/min until 40 % of the estimated capacity was reached. Then, the machine was kept at idle for 30 second and reduced to 10 % of F_{est}, where it was held at another 30 seconds of idle. Next, the specimen was ran till failure with the rate of 1.5 mm/min in order to keep control of the machine. This is slightly different from what is suggested in NS-EN 26891, but still believed to give reliable results as the data is processed digitally.

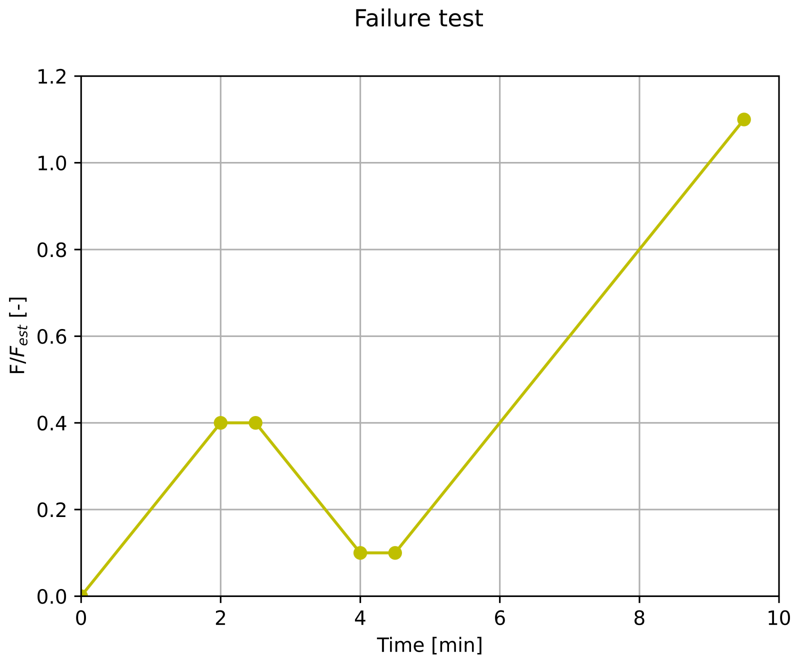
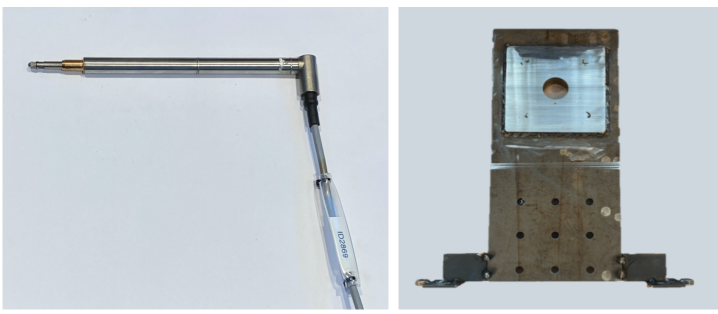


Figure 3.5: Failure test load procedure

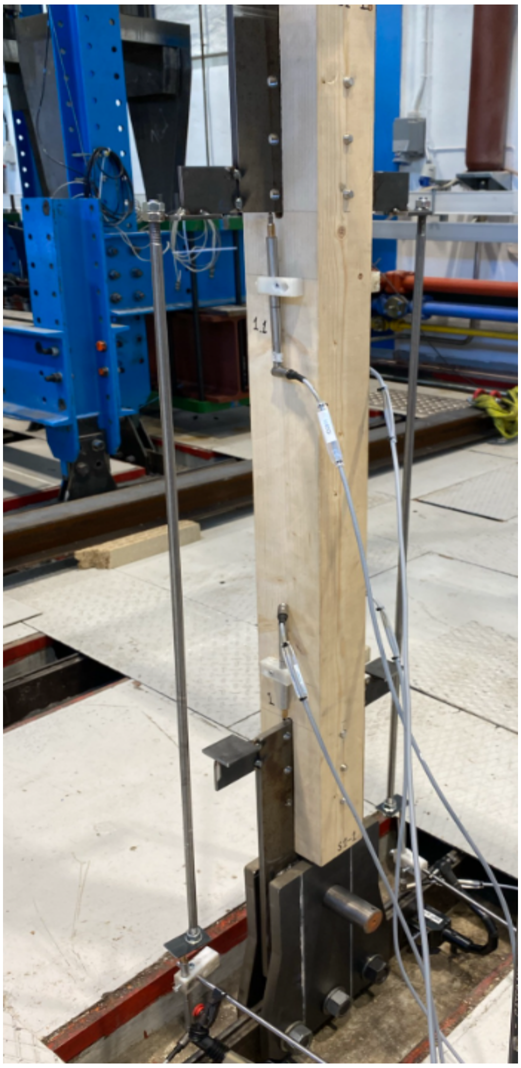
3.1.3 Setup

Linear Variable Differential Transformers (LVDTs), see figure 3.6a, were used in order to measure displacements between the slotted-in steel plate and wood. Instrumentation was adapted from Frette et al. (2021) to achieve comparable results. Two LVDTs were used in the bottom part of specimen type 3, while a total of four LVDTs were used for type 1 and 2. Two LVDTs were placed on each side of the specimen and denoted 1 and 2 in the bottom connection, while LVDTs in the top connection were denoted 1.1 and 2.2 correspondingly. The measured value from the LVDTs were averaged to make sure that the effect of rotation was taken into account. Instrumentation of the specimens is shown in figure 3.7. Extra steel was welded on each side of the steel plates to have a surface to measure the relative displacement between steel and wood, see figure 3.6b. The component measurement, denoted C1 and C2 on each respective side, were done through two straight metal rods hanging vertically from the top to the bottom steel plate. The instrumentation of the three specimens may be briefly seen in figure 3.7. More pictures and details of the setup are gathered in the digital appendix.



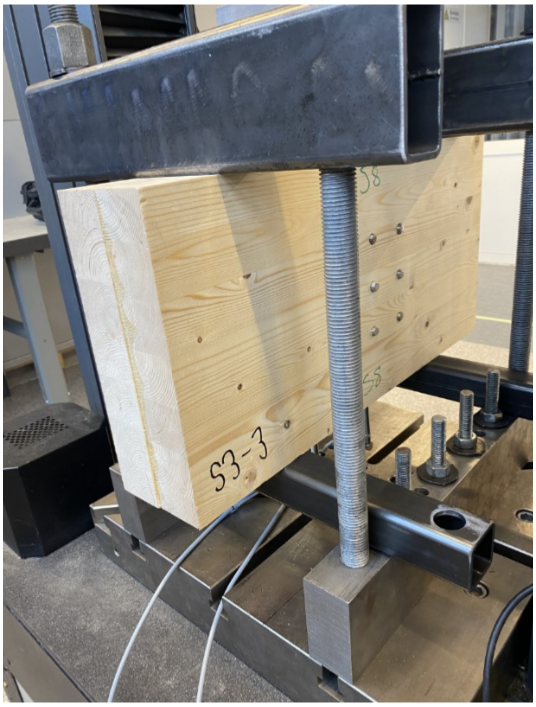
(a) LVDT (b) Steel plate

Figure 3.6: Setup equipment



(a) Type 1

(b) Type 2



(c) Type 3

Figure 3.7: Instrumentation of specimens

3.1.4 Notes of caution

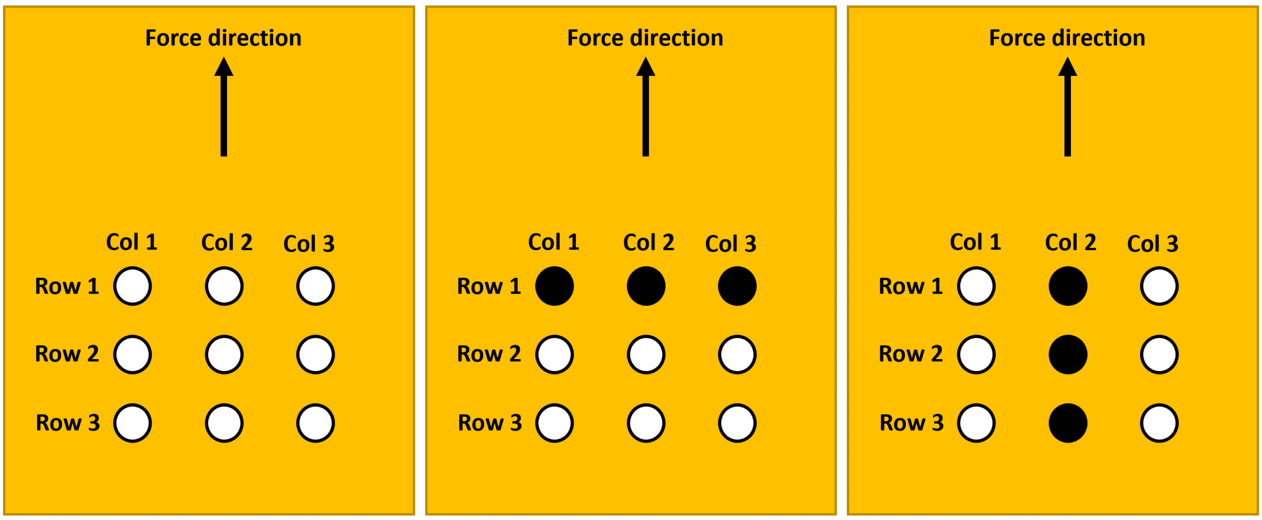
The following bullet points presents the most important notes of caution for the experimental work. In addition, a brief assumption of the incidents influence to the results is described. Furthermore, sources of error is discussed in section 5.2.

- For specimen type 3, an error was made during production, as described in Frette et al. (2021). The holes in the timber were predrilled with Ø13 instead of the planned Ø12, meaning the holes were slightly oversized. This should be kept in mind when interpreting the results, as the oversized holes probably gave somewhat more displacements than expected.

- When running the first test series for specimen type 3 (S3-1 and S3-2) the default setting of the logging program Catman was set to be only 3 decimals, resulting in small displacement changes not being logged due to an averaging error. When the error was discovered, the already performed test series were ran all over again, resulting in somewhat more usage of these specimens. As the maximum applied load level was limited to 40 % of F_{est}, it is believed to give minor influence to the results.

- When preparing testing of specimen type 1 and 2 in the 400kN actuator, it was discovered that the climate room, housing a temperature of 20 °C and relative humidity (RH) of 65 %, was out of order and currently at 17,9 °C and 35 % RH. When discovered, the specimens were immediately moved to another climate room with the desired temperature and RH. At that time, the specimens had been stored in the defect climate room for approximately three weeks. However, as the specimens originates from an earlier experiment and stored in the climate room in the mean time, the incident was probably not crucial to the results.

- In order to clarify what the authors have defined as rows and columns in a dowel-type connection, figure 3.8 was created. The definition is based on the same system used for matrices. This system is used as the basis for input in the numerical model, which is explained in detail in section 3.2. Examples are provided in figure 3.8b and figure 3.8c to illustrate how the different configurations are referred to in the present thesis. The system, i.e. rows horizontally orientated, is based on the force direction, meaning that it may be used for both parallel and perpendicular grain direction.



(a) Basis definition (b) Three dowels in one row (c) One dowel in three rows

Figure 3.8: Definition of rows and columns used in this thesis

3.1.5 Post-processing

Python scripts were made in order to handle the data from the experimental work and to produce plots visualizing the stiffness and energy dissipation in the specimens during testing.

3.1.5.1 Stiffness

The displacement measurement along the component was used to calculate the stiffness of the entire component, while the data collected from the LVDTs were the basis of connection stiffness calculations. LVDTs connected directly to the specimen provides higher accuracy than the component measurement, meaning that the component stiffness must be interpreted with somewhat more caution than the connection stiffness. However, for both stiffness calculations, the data was filtered such that only the last 6 cycles were included. For those cycles, a fitted line corresponding to the slope of the curves, represented the stiffnesses. The individual and averaged LVDT measurements and the corresponding fitted line is shown in figure 3.11a.

The component stiffness was calculated as described in section 2.4, i.e. as a sum of springs in a series. Equation (3.1) shows which elements that were included in the calculation:

$$K_{tot} = \left[\frac{1}{K_{bot}} + \frac{1}{K_{top}} + \frac{L_{wood}}{EA_{wood}} + \frac{1}{EA_{steel}}\right]^{-1} \quad (3.1)$$

where EA_{steel} and EA_{wood} is the axial stiffness for steel and wood respectively, and L_{wood} is the length of the specimen, measured from connection center-to-center, i.e. $L_{wood} = 880$ mm.

Stiffness of the specimens at load levels around zero were also calculated. The basis for such data was found from Frette et al. (2021) as fully reversed loading was not performed for the present thesis. The obtained data was ran through the same Python scripts in order to calculate stiffness in the area close to zero force and the full stiffness for the entire test series, as briefly illustrated in figure 3.9. Pure compression and tension stiffness calculations were performed already in the previous thesis by picking points along the compression and tension curves separately. Here, a linear fitted curve was found from all the measured data points representing the full stiffness, while a force threshold of ±5% of F_{max} were applied to find stiffness close to zero loading. The calculated stiffnesses are denoted K_{full} and K_{zero} respectively.

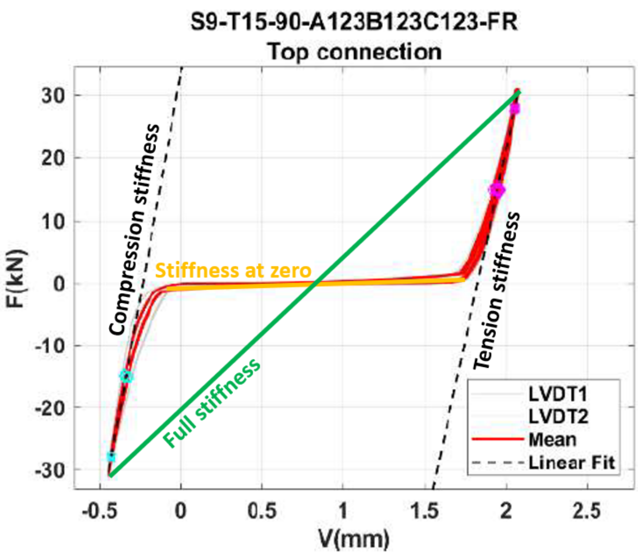
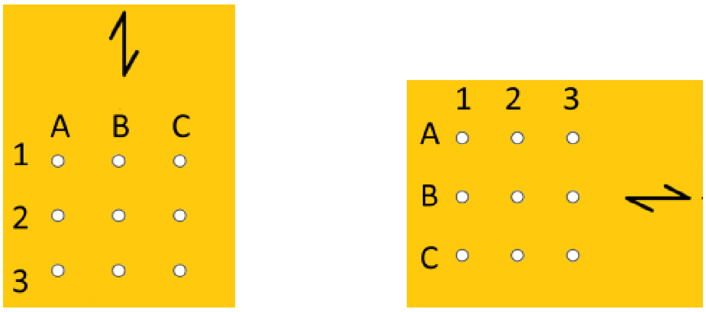


Figure 3.9: Illustration of stiffnesses calculated from data obtained by Frette et al. (2021).

In contrast to the present thesis, Frette et al. (2021) varied the dowel configuration in each specimen. All variations were analyzed in order to investigate how dowel configurations may contribute to the full stiffness and stiffness close to zero. The naming of the different configurations of the specimens were adapted from Frette et al. (2021) and follows from figure 1a and figure 1b for specimen S1/S2 and S3 respectively. For example, a configuration with one dowel in three rows is denoted B123, while three dowels in one row is denoted A1B1C1.



(a) S1 and S2 (b) S3

Figure 3.10: Configuration basis

In order to compare the measured and the calculated EC5 stiffness, and the degree of utilization of the connection for each load phase, two additional ratios, equation (3.2) and equation (3.3), were calculated.

$$R_{\text{stiffness}} = \frac{K_{\text{measured}}}{K_{\text{EC5}}} \tag{3.2}$$

$$R_{\text{utilization}} = \frac{F_{\text{max,i}}}{F_{\text{EC5}}} \tag{3.3}$$

where K_{measured} is the measured stiffness from each test and K_{EC5} is the calculated stiffness for the respective connection. $F_{\text{max,i}}$ is the maximum load in load phase i and F_{EC5} is the calculated capacity. The values are provided in table 3.1.

3.1.5.2 Energy dissipation

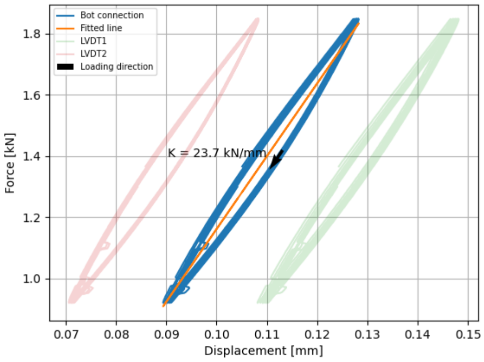
The energy dissipation was calculated from the hysteresis loops obtained in the cyclic loading, see figure 3.11a. A similar, but idealized, hysteresis loop is shown in figure 3.11b, and make up the basis for the sets of equations used in the energy dissipation calculations. As in figure 3.11a, the stress-based quantity (force, F) is placed on the vertical axis and deformation (displacement, u) on the horizontal axis. The change of elastic energy during *one* full cycle, ΔU_{max}, is calculated as (Kjell Arne Malo, 2021b)

$$\Delta U = \frac{1}{2}\frac{F_a^2}{k} \tag{3.4}$$

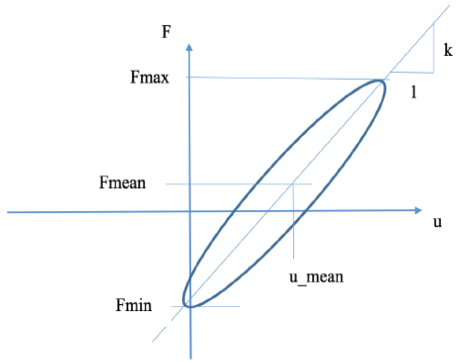
where k is the measured stiffness of each hysteresis loop. F_a is the amplitude of the applied load, given by equation (3.5).

$$F_a = \frac{1}{2}(F_{max} - F_{min}) \tag{3.5}$$

The dissipated energy, E_d, is then calculated as the averaged enclosed area of each included cycle. Finally, the equivalent viscous damping is given by:



(a) Loop obtained from lab data



(b) Idealized hysteresis loop (Kjell Arne Malo, 2021b)

Figure 3.11: Hysteresis loops av basis for stiffness and energy dissipation calculations

$$\xi = \frac{1}{4\pi} \frac{E_d}{\Delta U_{max}} = \frac{E_d \cdot k}{2\pi \cdot F_a^2} \tag{3.6}$$

3.1.5.3 Failure

For the failure test, the specimens were instrumented as described in section 3.1.3 and ran till failure according to figure 3.5. After failure, the specimens were photographed and visually investigated to determine the displayed failure mode. DIC recordings were performed on specimen type 3, but not directly utilized in the present thesis, but may be used for further work, see section 5.3.

Stress-strain (embedment stress-plastic strain) plots were created for each sample of all specimens according to theory presented in section 2.6. The ratio, $R_{failure}$, was introduced in order to investigate how the Eurocode estimates the capacity compared to the actual capacity, see equation (3.7).

$$R_{failure} = \frac{F_{failure}}{F_{est}} \tag{3.7}$$

The estimated capacities, F_{est}, is presented in table 3.1 and $F_{failure}$ is the maximum load before failure.

3.2 Numerical method

The Abaqus model has been developed with the intent of being fully parametric. This has been achieved using Python scripting for Abaqus. Almost all the features of the model are parametric and can be adjusted via the input in the Python script. One of the motivations to do this is to use an iterative approach to tune the Abaqus model to behave as the specimens in the lab and get accurate stiffnesses from the tests. And also to be able to use the model for different geometries than the ones that were tested in the lab. The script is written for Abaqus 2021, and the simulations have been run on a computer with 64 GB of RAM, four cores of 4 GHz and GPU acceleration using a Quadro P2000 graphics card.

The Python script is developed with a structure based upon Martin Pletz's 'Python script for Abaqus course' methodology (Pletz, 2022). The following functions are defined in order to generate the model, run the analysis and analyze the results:

- input_parameters – Can choose between the timber setups S1, S2, S3, Dorn or manually entered geometry
- make_geometry – Generating all parts, create partitions and generate mesh on each part
- make_sections – Assign section to each part with material and material orientation for wood
- make_assembly – Assemble the parts to form the connection
- make_boundaries – Create step for load application, apply load and boundary conditions, tie constraints and create the history output request
- run_model – Run the model and wait for completion
- evaluate_results – Create outputs from the ODB file: prints of stresses and deformation
- evalueate_historyOutput – Calculate average nodal displacement of top surface and the stiffness of the connection and add this to a result file

Further these functions are run in a new function defined for the whole model. This function can then be called upon in the code in order to run the Abaqus model and the input-variables can then be varied to see how they influence the stiffness of the connection. The full Python script can be found in appendix G.

3.2.1 Mesh

Most of the model has been modelled using HEX-elements of the type C3D8R: An 8-node linear brick, reduced integration, hourglass control element. In order to get a nice mesh with low distortion of the elements each part has been partitioned. The timber part block has been meshed with C3D4: A 4-node linear tetrahedron element. An element convergence study was performed in order to decide which parts should have a coarse or fine mesh. The study showed how the mesh size on each part influenced the connection stiffness, and the parts where meshed accordingly, see section 4.2.1. The regions with the highest stresses and change of stress were also meshed with a finer mesh.

3.2.2 Material model

The timber part is assigned the material model described in section section 2.5. With a material direction defined where the 1 direction follows the longitudinal axis of the part: the Y-axis for specimen S1 and S2 and for S3 the material direction follows the x-axis. The dowels have been assigned a homogeneous solid section with elastic and plastic yield strengths as given in the dowel certificate. In order to model the dowel-to-wood interaction an approach using two circular zones around the dowel has been applied. These regions are applied to avoid contact problems in the implicit analysis. The inner ring is modeled as a homogeneous solid section with a low stiffness and the outer ring has the same wood material model, but with a factor that reduces the stiffness compared to the rest of the wood. These parameters are tuned in order to achieve similar stiffness of the connection as those observed in the lab-tests.

3.2.3 Assembly

The assembly is composed of the following components:

- Steel plate
- Dowel
- Ring 1
- Ring 2
- Inner timber part
- Outer timber part

The Dowel, Ring 1, Ring 2, and inner timber part make up a basis block. As shown in figure 3.12e. The reason for using these two rings is to model the dowel zone without the use of contact. By giving the inner ring a low enough stiffness that ring can mimic the observed crushing of the wood in the tests, as seen by Dorn (2012).

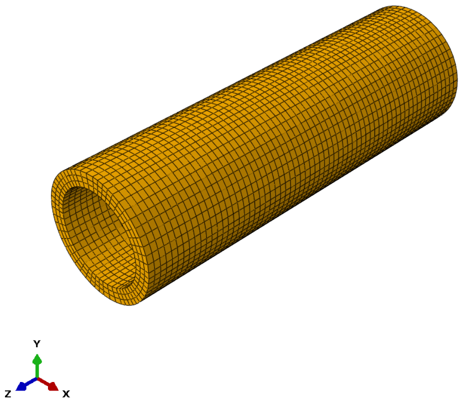
The outer timber part has a cut-out where one or more of these basis blocks are assembled depending on the dowel configuration that is chosen.

3.2.4 Boundary conditions

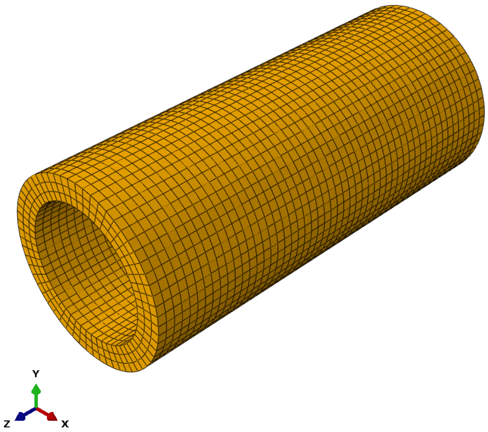
The boundary conditions in the model are created in the static step. As the model has been modelled taking advantage of the symmetry in the test specimens, to reduce computational time, there are symmetry boundary conditions on both plate, dowels and top of the wood part as can be seen in figure 3.14. The plate is also fixed in the bottom in the S1 and S2 configuration, as shown in figures 3.14a and 3.14b. While the figure 3.14c is constrained against movement in the red squares in the figure, corresponding to the hold downs used in the lab shown in figure 3.7c.

The different parts of the assembly are fastened together using "Tie constraints" as shown in figure 3.13. The dowel is tied either to the inner ring, figure 3.12b, or the timber block figure 3.12d directly, with the dowel as master-surface. Depending on the chosen dowel zone modelling strategy, the zone can be modelled in three different ways; using zero rings, one ring or two rings. The angle of the connection zone between the dowel and the next surface can also be specified, with angles ranging from 22.5 to 180 degrees. The reason this angle can be changed is to be able to mimic dowels of both low and high friction, with a corresponding low

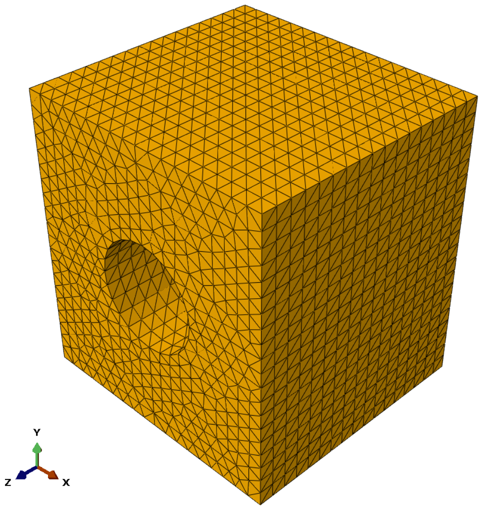
(a) Dowel



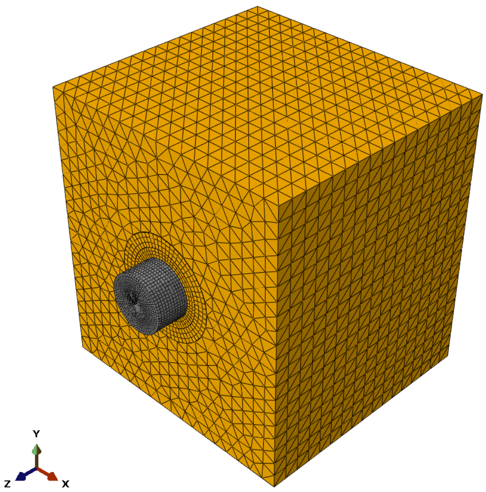
(b) Inner ring



(c) Outer ring



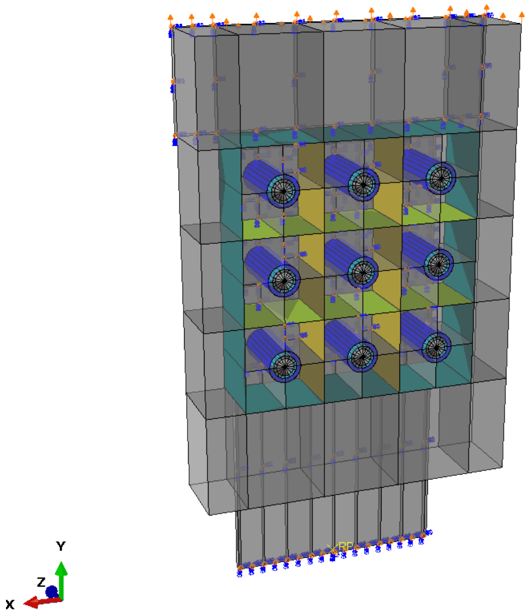
(d) Timber part



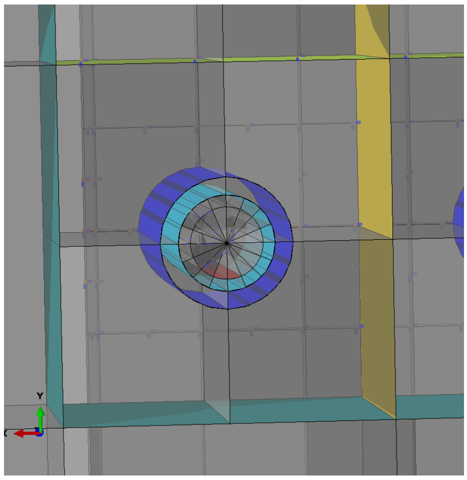
(e) Basis block

Figure 3.12: The parts that form a basis block

or high degree circle sector tie. The inner ring then ties to the outer ring, and the outer ring to the timber part. The sides of the timber part then either tie to other basis blocks or directly to the outer timber as can be seen in yellow or green respectively in figures 3.13a and 3.13b. The steel plate connect to the dowel using a 180 degree tie.



(a) Ties specimen S2



(b) Detailed view of one basis block

Figure 3.13: Tie constraints overview

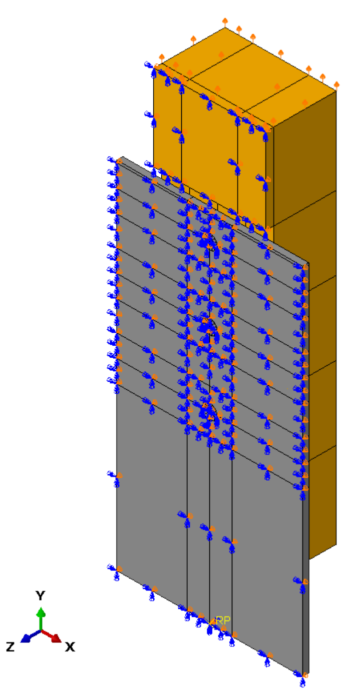
3.2.5 Analysis modes

The model has been programmed for two different analysis modes. The normal mode is meant to measure the stiffness of the model for a given prescribed load. This mode applies the load and measures the displacement to calculate the stiffness as the applied load divided by the measured displacement. The other mode is displacement driven, and is meant to simulate the failure test. Here a prescribed displacement is added to the model as shown in figure 3.14. For specimen S1 and S2 the displacement is applied in the top of the outer timber part as can be seen in figures 3.14a and 3.14b. S3 is implemented upside down, compared to the lab setup, for modelling simplicity and code reuse. The red squares seen in the figure correspond to the supports in the lab, and the displacement is prescribed from the plate end in negative y-direction.

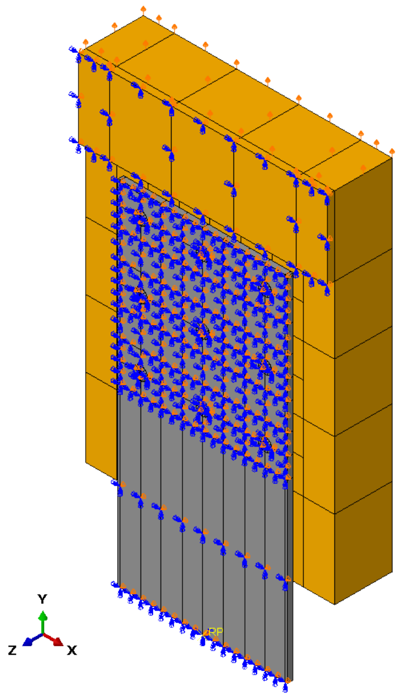
3.2.6 Post-processing of results

The tuning of the model has been done in order to get the same stiffness values from the Abaqus model as was measured in the lab-tests done by Frette et al. (2021) for a range of different setups. In order to achieve this, an automatic stiffness calculation has been implemented using the average nodal displacement U2 in the top surface in the connection model and the force the connection has been subjected to. These results have then been stored in a results array. And by iterating through different stiffnesses of the two rings, plots have been produced showing how the changed stiffnesses of the different parts of the model affect the connection stiffness.

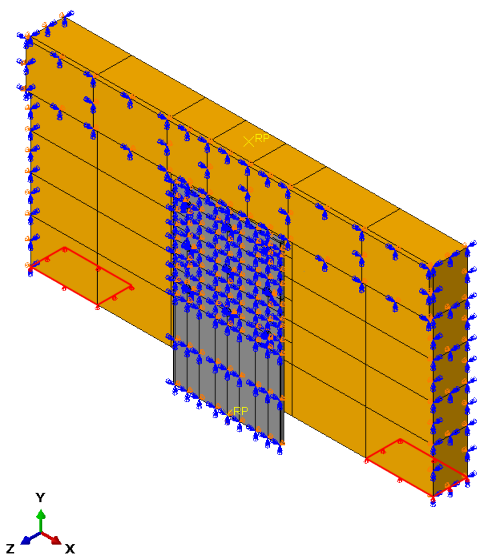
For a given setup, one optimization plot is produced with the corresponding stiffness in the inner and outer ring to achieve same stiffness in the Abaqus numerical simulation as in the lab test. Based on these plots a fitted stiffness has been produced taking into account number of fasteners in load direction when setting the stiffness of the rings.



(a) Load and boundary conditions S1



(b) Load and boundary conditions S2



(c) Load and boundary conditions S3

Figure 3.14: Loading and constraint of the test specimens in Abaqus

Chapter 4

Results and discussion

This chapter is, as chapter 3, separated into the experimental results (section 4.1) and numerical results (section 4.2). A discussion is included throughout this chapter in order to address strengths, weaknesses and make remarks about the presented results.

4.1 Experimental results

In this section, the results from the testing of each specimen are presented. The parameters introduced in section 3.1.5 are used in section 4.1.1 to section 4.1.3. All hysteresis loops obtained from the laboratory are included in appendix A to appendix C.

4.1.1 Stiffness

A total of seven different stiffness values are presented in the present thesis. $K_{measured}$ is the measured stiffness per shear plane per fastener and is presented alongside the calculated EC5-stiffness, here denoted K_{EC5}. $K_{measured,tot}$ and $K_{EC5,tot}$ represents the total measured and estimated stiffness of the respective connection where number of fasteners and shear planes are multiplied. These values are compared to the measured component stiffness, K_{comp}. Note that this only applies for specimen type 1 and 2, as a separate component stiffness for type 3 was not possible to measure due to its layout. $K_{measured,tot}$ is calculated according to equation (3.1).

For specimen type 1 and 2, $K_{measured}$ is calculated as the average of the top and bottom connection for all tested samples of the specimens. The process is illustrated in figure 4.1. For type 3, the process is similar, except from skipping the second and third step as there was no top and bottom connection to distinguish between.

In general, only the measured stiffness value for load phase 4 is compared to the results of Frette et al. (2021), as it was the only load phase with similar force magnitude.

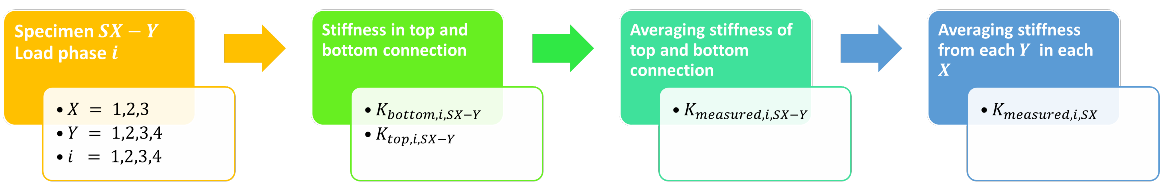


Figure 4.1: Work flow for calculating stiffness

Full and zero stiffness, as introduced in section 3.1.5.1, is presented for different configurations of the tested specimens. These stiffness values are only presented for load phase 4.

4.1.1.1 Stiffness | Specimen type 1

The measured stiffnesses for all load phases for specimen type 1 is displayed in table 4.1 and graphically illustrated in figure 4.2. The coefficient of variation in terms of equivalent stiffness, CoV, in figure 4.2, is shown in each of the four load phases as the error bar. The most obvious finding from the testing is that the measured stiffness was bigger than the calculated stiffness for all phases, where the maximum stiffness was measured to be 2,6 times higher. The best correspondence is found for load phase 4, but here the coefficient of variation is also calculated to be highest, resulting in slightly bigger variability in relation to the mean value. The obtained stiffness value for load phase 4 corresponds quite well to the measurements done by Frette et al. (2021) of 17,42 kN/mm, see table 2.2.

Table 4.1: Measured stiffness per shear plane per fastener for specimen type 1

Load phase	$K_{measured,S1}$ [kN/mm]	CoV [%]	K_{EC5} [kN/mm]
1	20.10	14.96	10.63
2	25.11	11.65	10.63
3	27.45	13.37	10.63
4	18.32	20.11	10.63

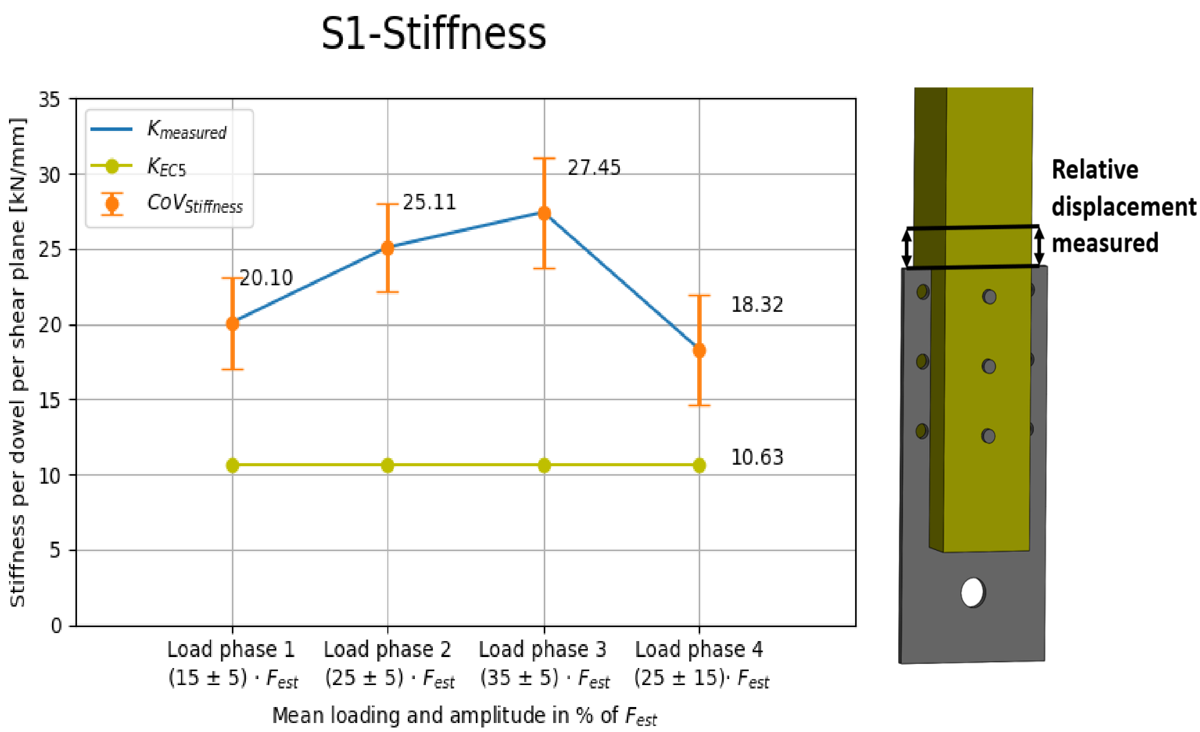


Figure 4.2: Test results stiffness specimen type 1

The recorded stiffness values for the component and the multiplied total connections stiffnesses are presented in table 4.2 and figure 4.3. It is evident from the results that there is good correspondence between the total EC5-stiffness and the measured component stiffness, while the measured total stiffness is well above. The component stiffness is, however, found to be bigger than the estimated stiffness for all load phases. Further discussion about the deviation between the component and measured total stiffness is given section 4.1.1.4.

Table 4.2: Measured total and component stiffness for specimen type 1

Load phase	$K_{measured,tot,S1}$ [kN/mm]	$K_{EC5,tot}$ [kN/mm]	$K_{comp,S1}$ [kN/mm]	CoV_{tot} [%]	CoV_{comp} [%]
1	56.12	30.69	33.81	14.96	6.90
2	68.91	30.69	41.29	11.65	14.49
3	74.76	30.69	51.03	13.37	3.72
4	51.47	30.69	35.83	20.11	3.72

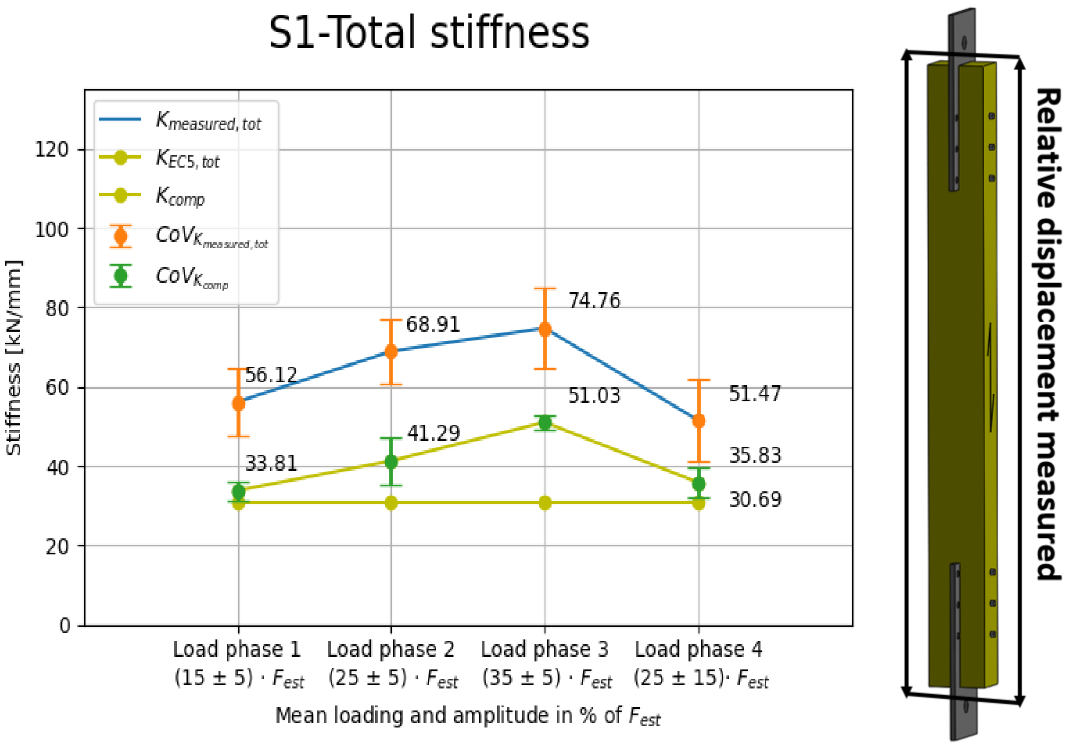


Figure 4.3: Test results total stiffness specimen type 1

Results for full and zero stiffness are presented in table 4.3 and displayed in figure 4.4. The different configurations with the corresponding stiffness values are included in the figure. The same tendency is evident for both K_{full} and K_{zero}, displaying a higher stiffness per dowel per shear plane for dowels in several rows. Simultaneously, the coefficient of variation is a lot higher for the zero stiffness than the full stiffness.

Table 4.3: Measured zero and full stiffness per shear plane per fastener for specimen type 1

Configuration	$K_{zero,S1}$ [kN/mm]	$K_{full,S1}$ [kN/mm]	CoV_{zero} [%]	CoV_{full} [%]
B1	0.20	3.36	5.20	6.37
B12	0.57	3.88	105.31	34.66
B123	0.38	3.38	85.67	20.52

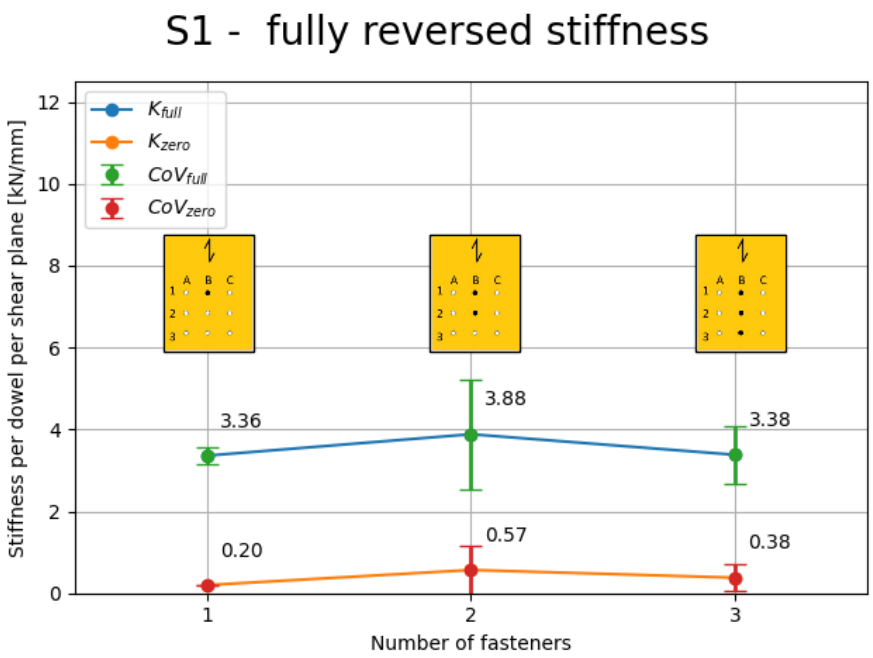


Figure 4.4: Stiffness for different configurations of specimen type 1

Figure 4.5 shows the individual results obtained from the fully reversed cycle for S1-1. Note that the value for B123 presented in table 4.3 is an average of the top and bottom connection for S1-1 to S1-3, but this is included to give an impression of the individual data. All obtained loops are included in appendix C.

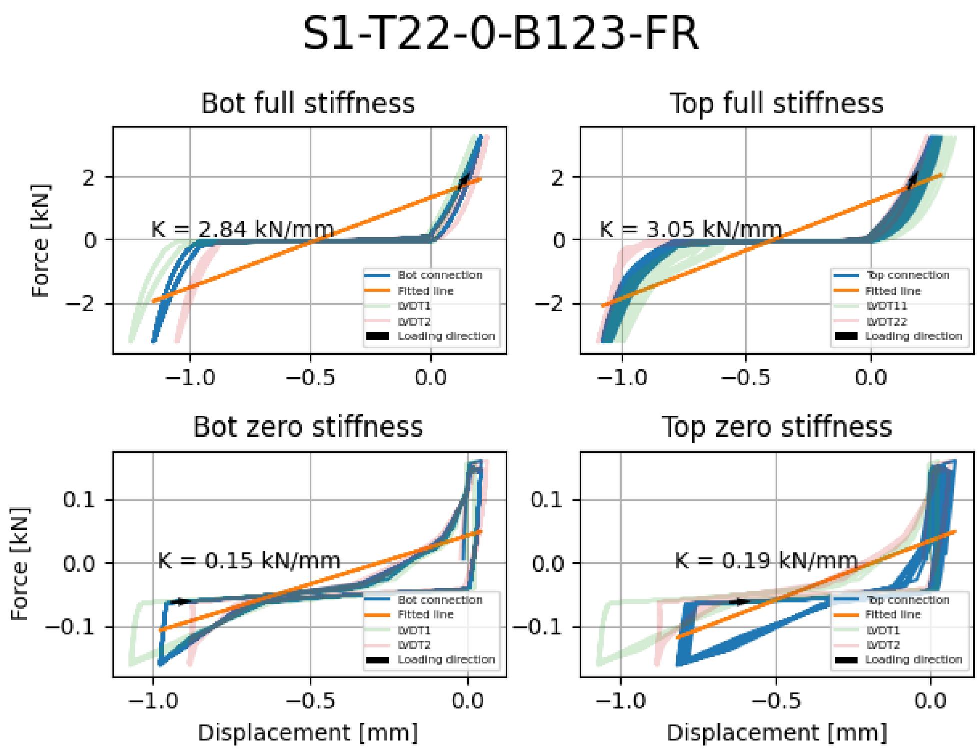


Figure 4.5: Results from analysis of S1-1 configuration B123.

4.1.1.2 Stiffness | Specimen type 2

Test results for specimen type 2 is presented in table 4.4 and illustrated in figure 4.6. Note that the Eurocode stiffness differs from type 1, as the density for the two specimens were different. The highest stiffness value recorded was 17.47 kN/mm, which is nearly twice as big as the estimated Eurocode stiffness of 9.30 kN/mm.

Table 4.4: Measured stiffness per shear plane per fastener for specimen type 2

Load phase	$K_{measured,S2}$ [kN/mm]	CoV [%]	K_{EC5} [kN/mm]
1	11.02	21.46	9.30
2	15.48	14.75	9.30
3	17.47	12.67	9.30
4	11.57	15.02	9.30

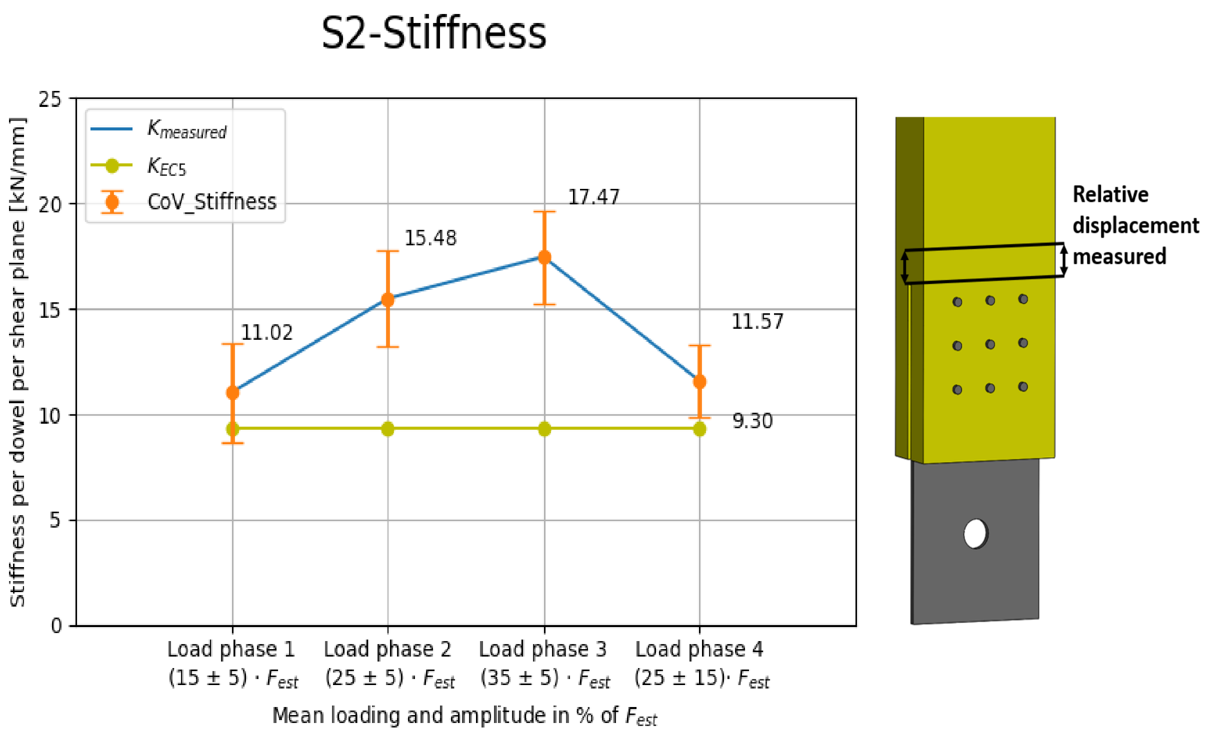


Figure 4.6: Test results stiffness specimen type 2

Table 4.5 and figure 4.7 presents the component measurements for specimen type 2. Qualitatively, the same results as for specimen type 1 (figure 4.3) is observed. In general, quite low CoV-values were calculated for the component measurements here.

Table 4.5: Measured total and component stiffness for specimen type 2

Load phase	$K_{measured,tot,S2}$ [kN/mm]	$K_{EC5,tot}$ [kN/mm]	$K_{comp,S2}$ [kN/mm]	CoV_{tot} [%]	CoV_{comp} [%]
1	95.43	82.67	74.15	21.46	11.96
2	135.19	82.67	103.30	14.75	6.74
3	153.02	82.67	115.37	12.67	5.81
4	101.62	82.67	78.46	15.02	5.81

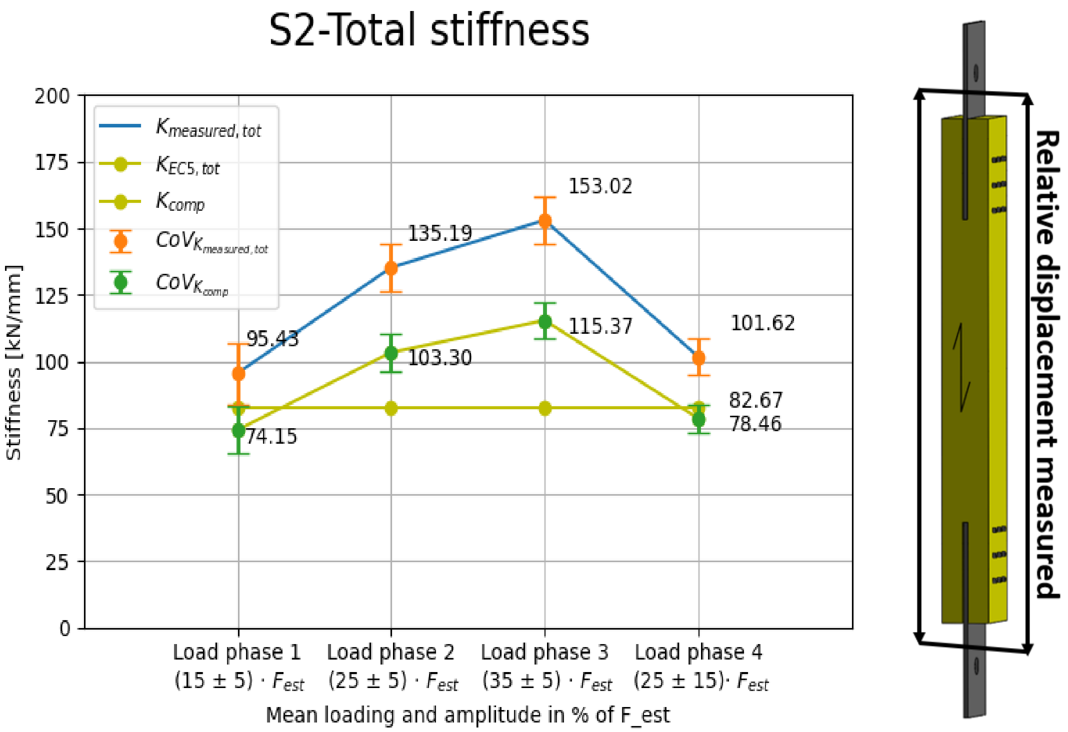


Figure 4.7: Test results total stiffness specimen type 2

Results for full and zero stiffness is presented in table 4.6 and displayed in figure 4.8. A total of 7 different configurations were tested. For configurations with 4 fasteners or more, the full and zero stiffness tends to converge to a certain value. For three dowels, the stiffness is higher for three dowels in the same row, than one dowel in three rows. This applies for both full and zero stiffness. An interesting observation may be done on configuration B1, where K_{zero} is found between 2,6 and 5,4 times bigger than all the other values. No big outliers were found within the data set and according to Frette et al. (2021) the slip was found to be highest for one dowel, intuitively meaning the stiffness should have been lower for this measurement. The uncertainties regarding where to delimit the data may, however, be the reason why the value is surprisingly high.

Another important remark to keep in mind is that the coefficient of variation in general is found to be quite high for these measurements.

Table 4.6: Measured zero and full stiffness per shear plane per fastener for specimen type 2

Configuration	$K_{zero,S2}$ **[kN/mm]**	$K_{full,S2}$ **[kN/mm]**	**CoV$_{zero}$ [%]**	**CoV$_{full}$ [%]**
B1	2.43	3.08	29.25	10.33
B12	0.66	3.13	84.25	22.50
B123	0.45	3.19	40.03	17.27
A1B1C1	0.65	4.34	44.95	9.08
A13C13	0.88	4.40	59.16	17.77
A12B12C12	0.87	4.23	42.89	16.03
A123B123C123	0.94	4.41	33.40	12.80

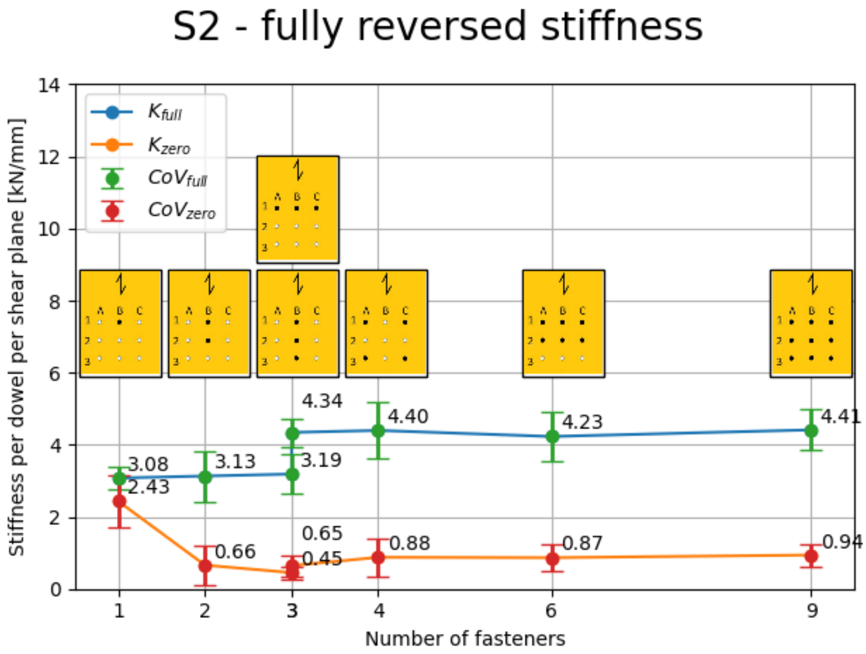


Figure 4.8: Stiffness for different configurations of specimen type 2

The individual results for specimen S2-2 configuration A13C13 (denoted S5 according to Frette et al. (2021)), is shown in figure 4.9. This particular result is included in order to illustrate an important aspect with these calculations. Even though the full cycles in the top and bottom connection seemingly look the same, they display quite different stiffnesses, showing the importance of not concluding or interpreting from single results.

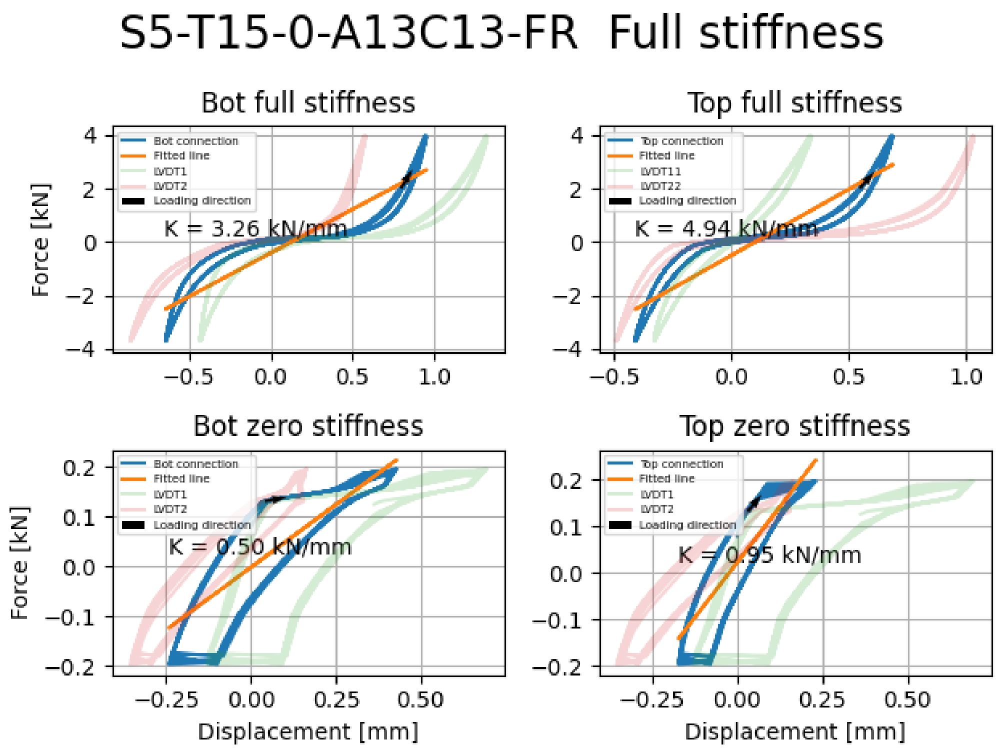


Figure 4.9: Results from analysis of S2-2 configuration A13C13.

4.1.1.3 Stiffness | Specimen type 3

Test results are provided in table 4.7 and illustrated in figure 4.10. In general, the estimated EC5 stiffnesses tend to correspond better with the measured data for specimen type 3. For load phase 1,2 and 4, the measured stiffnesses are lower than the estimated, meaning EC5 being non-conservative. Somewhat higher values were recorded for test sample S3-1 compared to S3-3 for all load phases (values provided in appendix A), but none of the deviations were considered to be disproportionately big and consequently none had to be removed. The CoV is found nearly similar for all load phases.

Table 4.7: Measured stiffness per shear plane per fastener for specimen type 3

Load phase	$K_{measured,S3}$ [kN/mm]	CoV [%]	K_{EC5} [kN/mm]
1	5.53	10.50	9.31
2	8.02	10.56	9.31
3	9.88	11.04	9.31
4	7.34	11.20	9.31

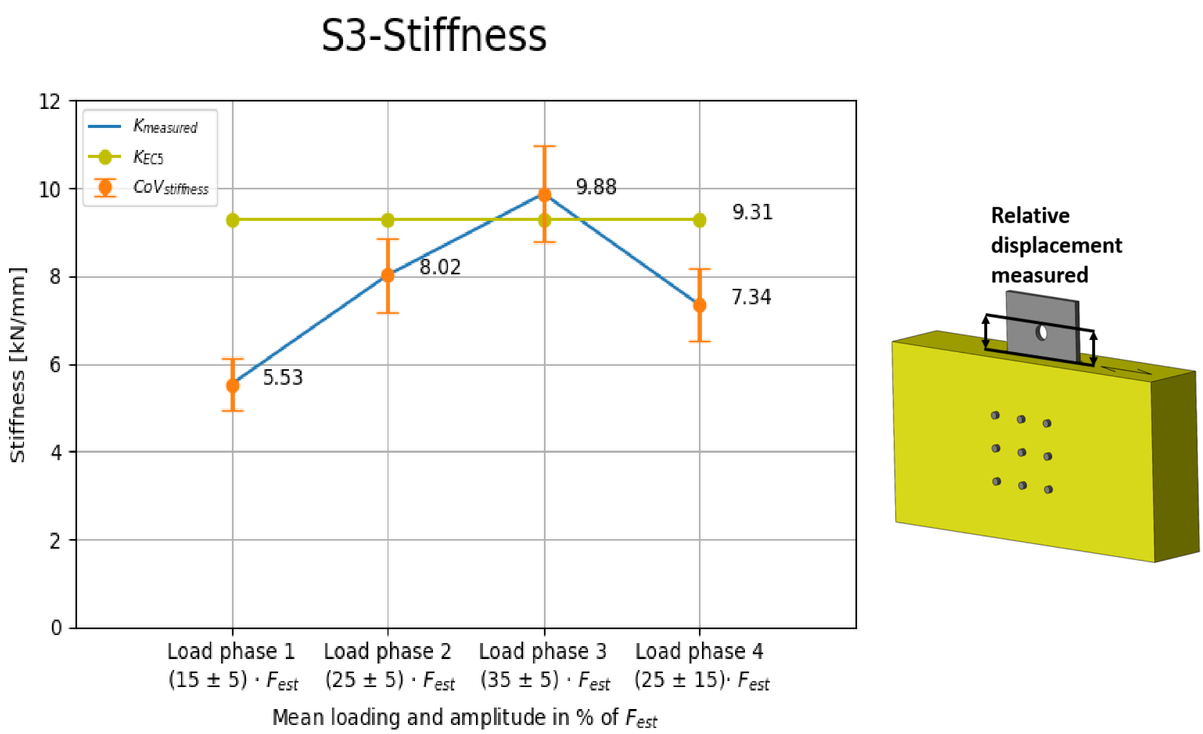


Figure 4.10: Test results stiffness specimen type 3

Results for K_{full} and K_{zero} is presented in table 4.8 and displayed in figure 4.11. 6 different configurations were tested for specimen type 3. The zero stiffness tends to be constant for all configurations, except from for three dowels in three different rows. As an opposition to S2, the stiffness per fastener per shear plane tends to decrease for an increased number of fasteners.

Table 4.8: Measured zero and full stiffness per shear plane per fastener for specimen type 3

Configuration	$K_{zero,S3}$ [kN/mm]	$K_{full,S3}$ [kN/mm]	CoV_{zero} [%]	CoV_{full} [%]
B2	0.09	1.46	3.71	0.71
A2C2	0.09	1.45	7.16	3.96
B123	0.10	1.46	4.02	2.49
A2B2C2	0.05	0.66	32.77	9.23
A123C123	0.09	1.16	21.74	4.29
A123B123C123	0.07	0.86	24.04	6.23

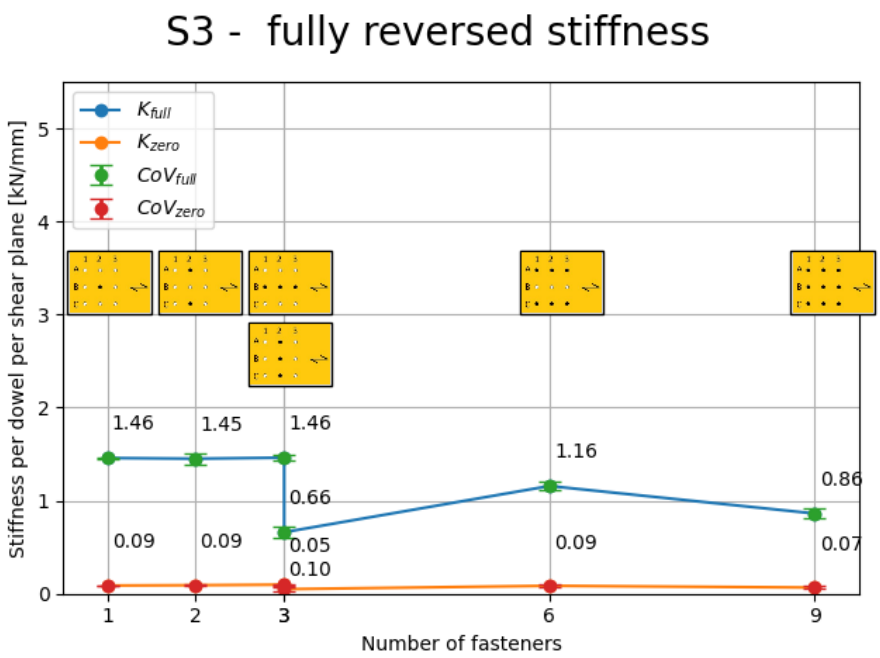


Figure 4.11: Stiffness for different configurations of specimen type 3

The individual results for S3-3 (originally S9) is displayed in figure 4.12. For the zero stiffness, it can be seen that the force limit might be a bit high, giving some uncertainties regarding what should be considered to be around zero loading.

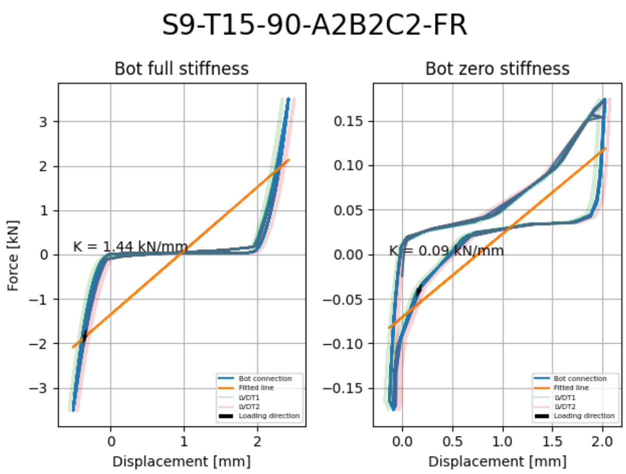


Figure 4.12: Results from analysis of S3-3 configuration A2B2C2.

4.1.1.4 Stiffness | Comparison and remarks

The following observations are done with a high level of consistency:

- For the stiffness measurements, the results show the same tendency; a clear increase in stiffness for higher loading, see figure 4.13. The stiffness drops for all specimens in load phase 4, which operates on the largest loading interval with greatest difference between minimum and maximum loading. $R_{stiffness}$-ratio (defined in equation (3.2)) of 1 indicates that the measured stiffness coincides perfectly with the theoretical estimated stiffness. The best correspondence is found for specimen type 3 in load phase 3, i.e. loading $35\% \pm 5\%$ of F_{est}, while the worst is found for specimen type 1 in the same load phase. Here, the measured stiffness is found to be more than 2,5 times higher than the estimated, meaning the stiffness of the connection is underestimated by the Eurocode.

- Higher stiffness for higher maximum loading implies that load level possibly could have been included as a part of the Eurocode-calculation of stiffness. A possibility could, for instance, be a factor similar to k_{mod}, which takes load duration and climate class into account. With such a factor, one could take advantage of connections seemingly displaying higher stiffness for higher utilization. This would potentially allow more efficient design of connections and structures, which can contribute to reach the climate goals of the construction sector, as briefly mentioned in chapter 1. On the other hand, as long as the actual stiffness is higher than the calculated, the calculations are conservative and thus on par with the rest of the Eurocode-rules. Anyway, these results verify the findings of Sandhaas et al. (2020), concluding that stiffness is strongly dependant on load level.

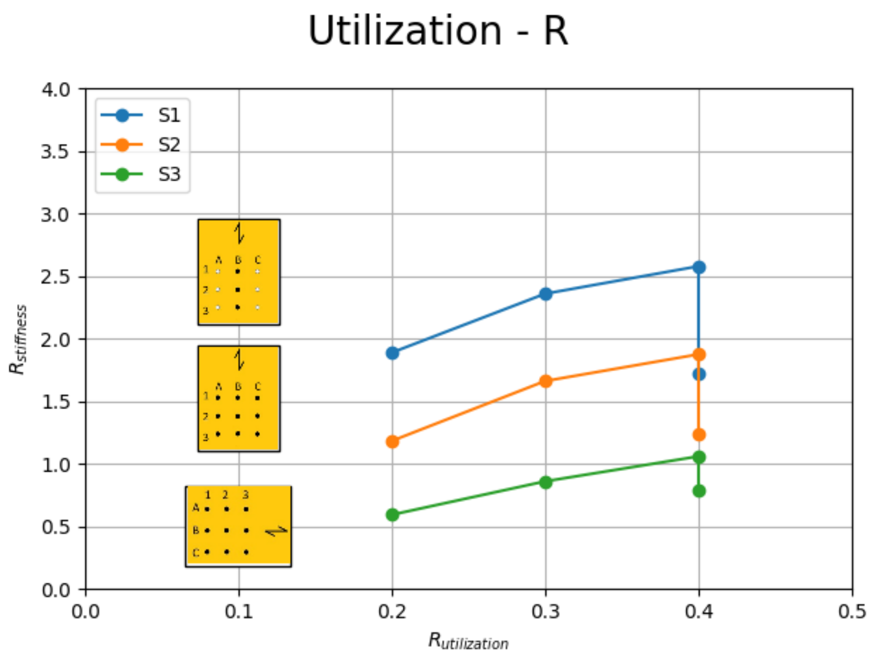


Figure 4.13: Comparison of stiffness measured in specimens

- EC5 underestimates the stiffness during along-the-grain-loading (specimen type 1 and 2), i.e. the measured stiffness is higher than the estimated. The correspondence is, in general, better for loading perpendicular to grain (specimen type 3).

- When the data series of Frette et al. (2021) were re-analyzed, it was discovered that the stiffness measured for specimen type 3, sample 3 (denoted S9 in the previous thesis), were somewhat higher than for the two other samples. The same observation was done in the present thesis, giving increased reliability to the results. The measured stiffness did, however, not correspond perfectly, but might be due to some small differences in how the LVDTs were attached to the specimens. These seemingly small errors may influence the results and add uncertainties.

- Distinctly lower stiffness values were recorded for specimens loaded perpendicular to grain, indicating that angle to grain should have been a parameter included in the stiffness equation, see equation (2.10).
- A very small stiffness value is recorded for fully reversed loading close to zero load level. This shows that there is some kind of stiffness, despite that slip values up to 2.31 mm have been recorded (Frette et al., 2021).
- The stiffness calculated for the whole fully reversed cycle is, without exceptions, lower than the stiffness obtained from the tension and compression part separately. This parameter is quite interesting as dynamic loads induced by machinery may create oscillations around zero loading resulting in an alternating load sign. Based on the findings in the present thesis, connections exposed to fully reversed loading may display a smaller stiffness than calculated, potentially resulting in larger vibrations and deflections in service. Nevertheless, it is important to add that the usual load situation is such that the structure is either loaded in tension or compression, and then exposed to dynamic loads. Taking this into consideration, the measured stiffness values for the load procedure presented in section 3.1.2 should be given higher importance than the zero and full stiffness.
- As described in section 3.1.4, there were some uncertainties whether the moisture content (MC) was constant in the specimens or not due to the climate room error. The MC was measured before testing for each specimen and the results are provided in table 4.9. As can be seen, the MC tends to be quite constant, indicating that the influence might not be too big.

Table 4.9: Measured MC in specimens before testing

Specimen	S1-1	S1-2	S1-3	S2-1	S2-2	S2-3	S2-4	S3-1	S3-2	S3-3
MC [%]	-	10,8	10,7	10,9	12,7	12,9	12,7	10,9	11,1	11,6

- The difference between the component measurement and the total stiffness calculations (see equation (3.1)) is that not all contributions to the component stiffness are included. The reason for this lies in the placement of the LVDTs, not taking the contribution of the entire steel plate into account. Additionally the component measurement should be interpreted with caution as it was difficult to ensure a way to measure the total displacement without adding sources of error. It was challenging to keep the rods completely straight along the component during testing, and any small errors may affect the results.

4.1.2 Energy dissipation

Energy dissipation is presented in terms of the average calculated viscous damping ratio, ξ_{SX}. The same procedure as described in section 4.1.2 and figure 4.1 is applied, i.e. ξ is the average of the top and bottom connection for the tested samples. The Eurocode for timber bridges states that the viscous damping of structures with mechanical joints can be taken as 0.015 (CEN, 2004a). EC5 does however not provide a framework for estimating viscous damping in connections, but values from 0.05-0.07 can be found in literature (Pousette, 2001).

4.1.2.1 Energy dissipation | Specimen type 1

In table 4.10 and figure 4.14 the damping results for specimen type 1 is presented. The coefficient of variation for all load phases is quite high and may be explained by the measurements for the top and bottom connection for S1-1 and S1-2 deviated with a factor of more than 2. This should be kept in mind when interpreting the results. The results are in the range of 7,7 and 10,7 %, which is quantitatively slightly higher than the results obtained by Frette et al. (2021). The recorded viscous damping for load phase 4 in the present thesis is, however, very similar to $\xi = 0,073$, as previously recorded (Frette et al., 2021).

Table 4.10: Damping values S1

Load phase	ξ_{S1} [-]	$\xi_{S1,comp}$ [-]	$\mathbf{CoV}_{\mathbf{Damping}}$ [%]	$\mathbf{CoV}_{\mathbf{Damping,comp}}$ [%]
1	0.101	0.104	35.9	40.9
2	0.086	0.117	35.5	55.4
3	0.107	0.116	36.5	40.0
4	0.077	0.062	30.7	34.0

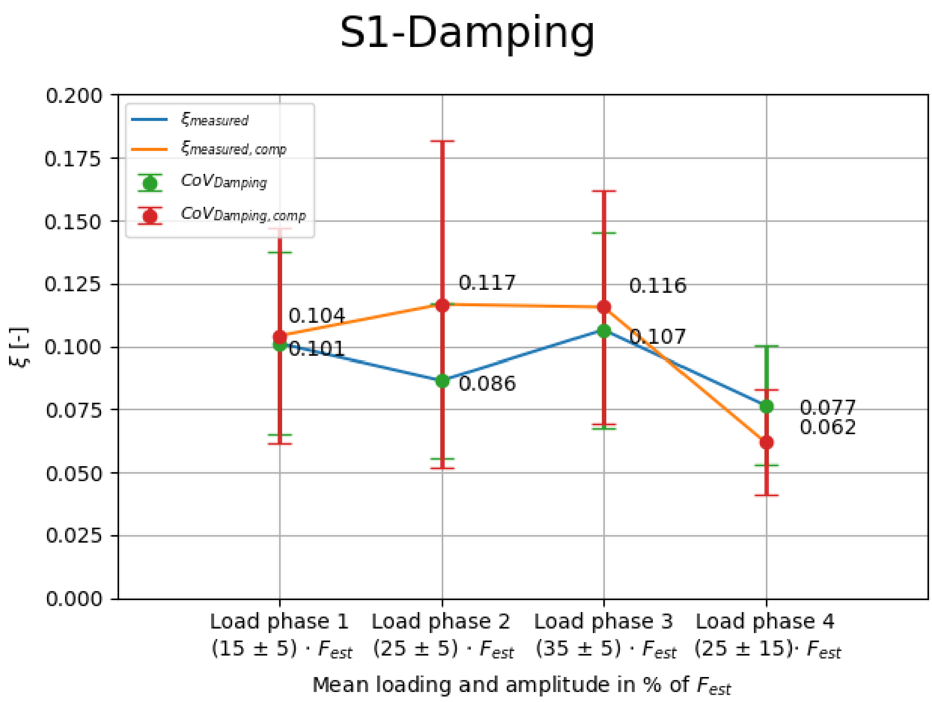


Figure 4.14: Damping results S1

4.1.2.2 Energy dissipation | Specimen type 2

In table 4.11 and figure 4.15 the damping results for specimen type 2 are presented. The coefficient of variation is considerably higher for load phase 2 and 3, but no big outliers are observed in the data set. Exactly the same viscous damping ratio in load phase 4 is found in the present thesis as by Frette et al. (2021), giving increased reliability to the results.

Table 4.11: Damping values S2

Load phase	ξ_{S2} [-]	$\xi_{S2,comp}$ [-]	**CoV$_{\text{Damping}}$** [%]	**CoV$_{\text{Damping,comp}}$** [%]
1	0.097	0.141	12.8	20.1
2	0.104	0.162	25.7	18.6
3	0.156	0.208	22.0	13.9
4	0.070	0.078	13.8	19.1

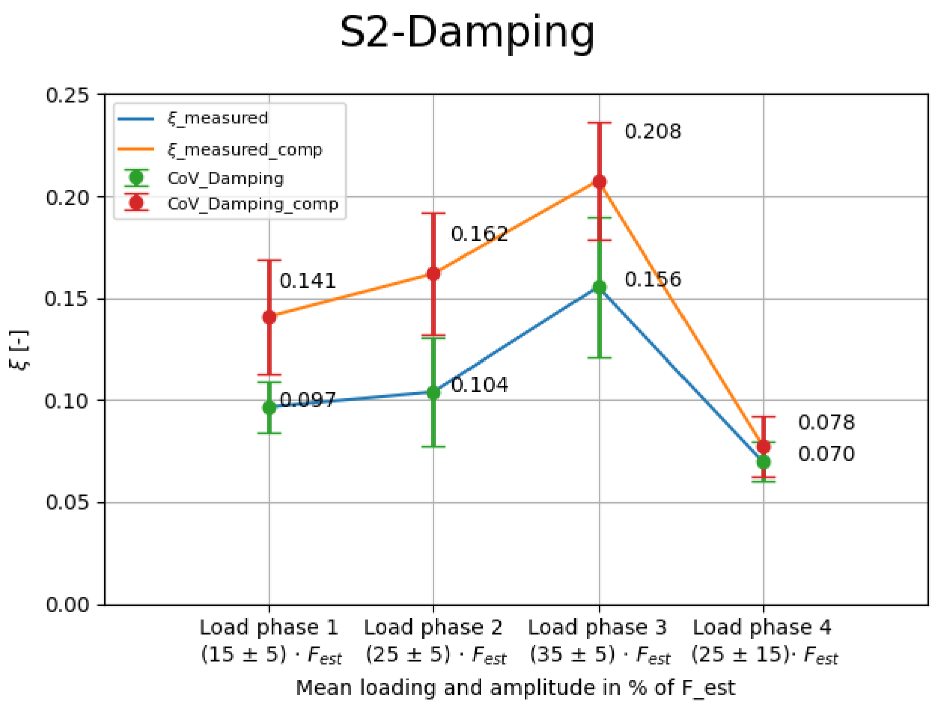


Figure 4.15: Damping results S2

4.1.2.3 Energy dissipation | Specimen type 3

In table 4.12 and figure 4.16 the damping results for specimen type 3 are presented. There was a huge difference in the measured damping value between S3-1 and S3-3, where the lowest value was 2,3 times smaller than the highest. This is reflected through the high CoV-values, especially for load phase 2 and 3. The recorded values for S3-1 was in general lower than the two other specimens, but considered to be nothing else than natural variations in wood. $\xi = 0.029$ (Frette et al., 2021) indicates good correspondence with the value obtained in the present thesis.

Table 4.12: Damping values S3

Load phase	ξ_{S3} [-]	$\text{CoV}_{\text{Damping}}$ [%]
1	0,015	15,0
2	0,014	30,4
3	0,020	38,7
4	0,023	11,5

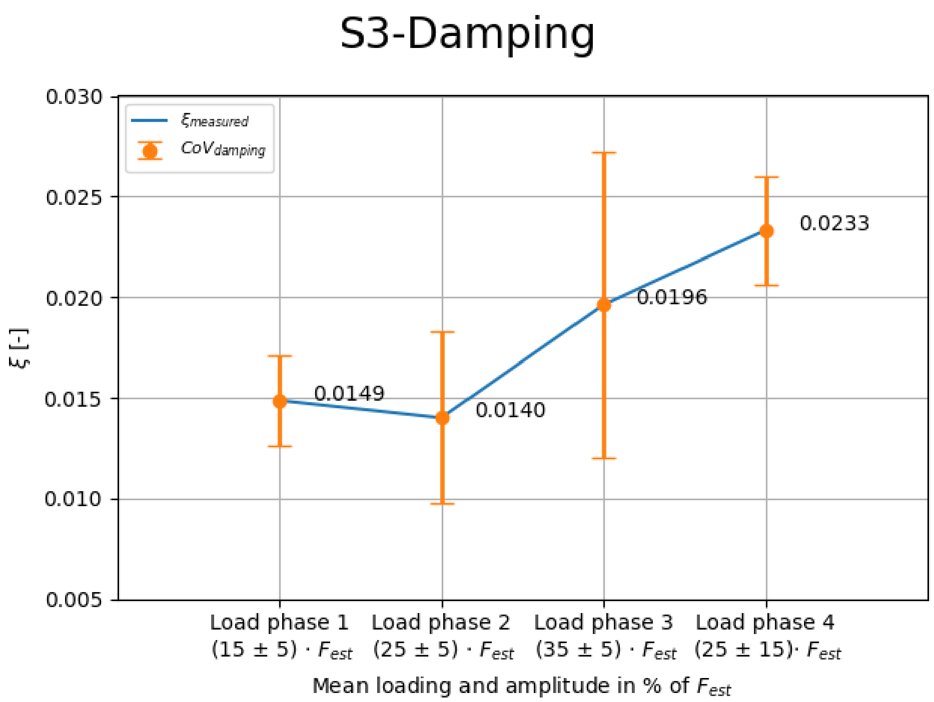


Figure 4.16: Damping results S3

4.1.2.4 Energy dissipation | Comparison and remarks

The following remarks must be kept in mind when interpreting the results:

- The equivalent viscous damping coefficient says something about the energy dissipation in the connection. However, in timber design, the total damping of the system is far more interesting than the single damping coefficient in each connection.
- As for stiffness, a clear tendency may not be found for damping. In fact, no clear tendency applying for all three specimens may be found at all.
- The measured damping values are in general a lot higher for loading parallel to grain (S1 and S2) compared to perpendicular to grain (S3).
- Compared to the suggested Eurocode value of $\xi = 0.015$ for structures with mechanical joints, the best correspondence is found for specimen type 3, as the measured values for type 1 and 2 are in general too high. The values for S1 and S2 coincide better with the values found in literature for timber connections, but one should be aware that these values origins from 2001 (Pousette, 2001).
- For all measurements except for one (S1, load phase 4, see figure 4.14), the viscous damping of the entire component is found to be higher than the connections separately. This means that some energy is absorbed in the cross section of wood directly. The difference is, however, not disproportionately large, implying that most of the energy is absorbed in the connections.
- The same uncertainties regarding the viscous damping for the component measurement applies as described in section 4.1.1.4.
- Zonta et al. (2011) stated that very few experimental works dealing with damping in timber structures only exist. For those works that were investigated and compared, it proved to be challenging to define rules that applies equally for any timber structure. The investigated results were strongly dependant on type of connections, which is evident from the results obtained in the present thesis as well. Hence, the results from S1/S2 and S3 may not be comparable and used as a basis for general rules.

4.1.3 Failure

Parameters introduced in section 3.1.5.3 is presented here for each specimen. Where it was possible(S1 and S2), the top and bottom connection was processed separately, since the failure, as expected, did not occur in both connections simultaneously. $\Delta u_{failure}$ denotes the displacement at maximum applied load. To obtain plastic strains, a connection length of $L = 120$ mm and $L = 110$ mm for S1/S2 and S3 respectively was used. Forces were transformed to embedment stresses by dividing by the dowel-to-timber contact surface, i.e. $A = d \cdot l$.

Several photographs of the failure were taken, but only a limited number is included in the thesis. All photos may, however, be seen in the digital appendix.

4.1.3.1 Failure | Specimen type 1

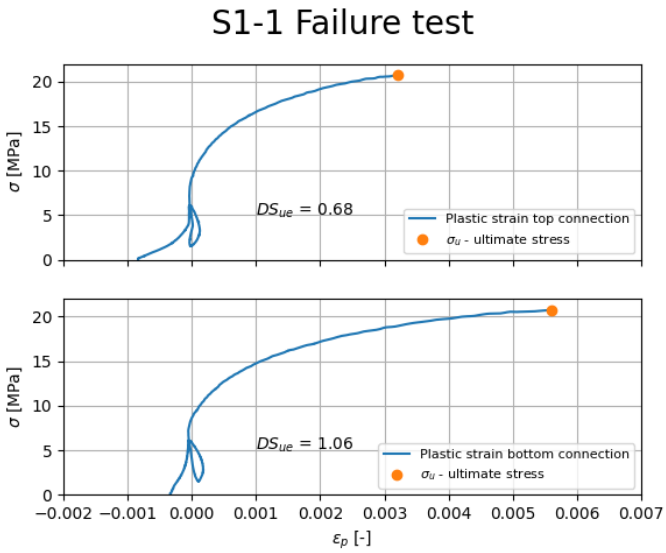
Failure curves for three samples of specimen type 1 are shown in figure 4.18. Failure values are provided in table 4.13. The highest recorded force was 76.3 kN, while the lowest was about 10 kN smaller. The $R_{failure}$ ratio varied between 1.20 and 1.37, resulting in an average of 1.28. The ductility, Ds_{ue}, for S1-2 was calculated to be considerably lower than for the two others. The reason for this is not clear to the authors. The actuator was ran until splitting parallel to grain occurred to both sides on all samples. Photos of the failure is shown in figure 4.17.

(a) Failure of S1-1, side 1 (b) Failure of S1-1, side 2

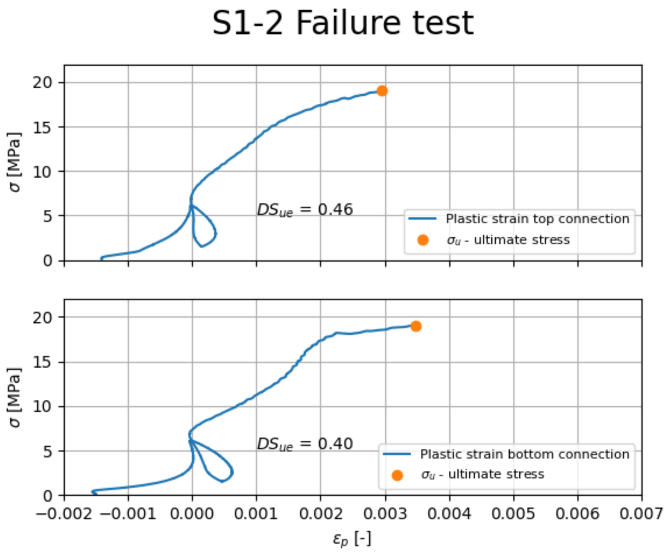
Figure 4.17: Failure of specimen type 1

Table 4.13: Failure values for specimen type 1

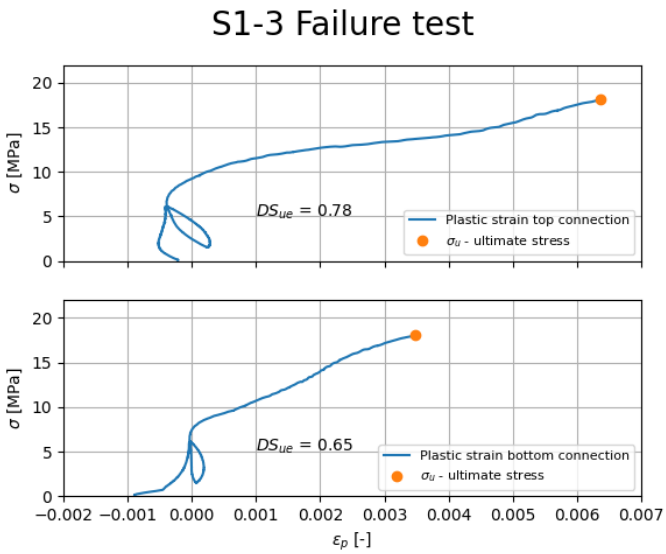
Specimen	Connection	$\Delta u_{failure}$ [mm]	$F_{failure}$ [kN]	$R_{failure}$ [-]	Ds_{ue} [-]	ϵ_i [-]
S1-1	Top	1.05	76.3	1.37	0.677	0.00084
	Bottom	1.35	76.3	1.37	1.058	0.00034
S1-2	Top	1.28	69.9	1.26	0.463	0.00141
	Bottom	1.62	69.9	1.26	0.403	0.00156
S1-3	Top	1.77	66.4	1.20	0.778	0.00052
	Bottom	1.17	66.4	1.20	0.648	0.00089



(a) S1-1



(b) S1-2



(c) S1-3

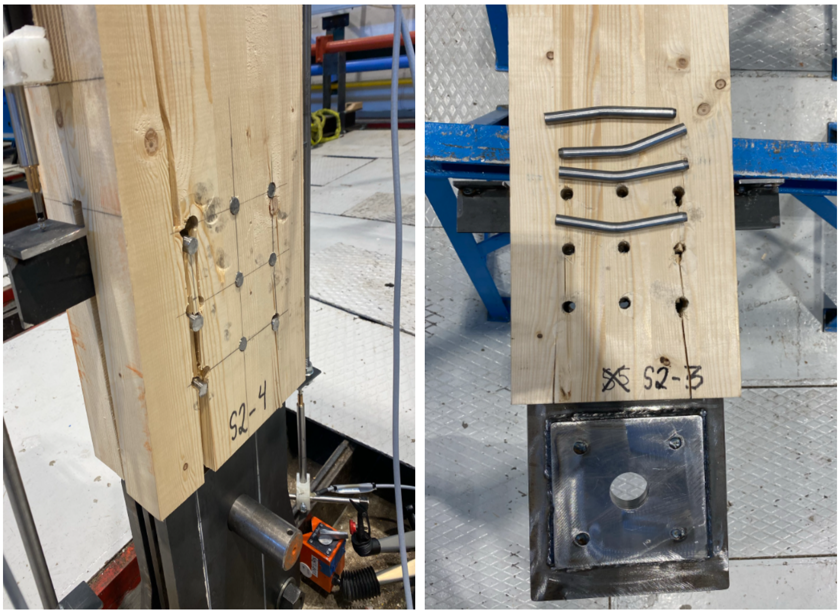
Figure 4.18: Failure curves S1

4.1.3.2 Failure | Specimen type 2

The failure curves for the four specimen type 2 samples are displayed in figure 4.20. The corresponding force and displacement values and calculated ductilities are provided in table 4.14. The failure curves in general look quite similar for all samples, except from S2-1, where the load drops at a level lower than the estimated capacity, see figure 4.20a. This indicates some sort of failure in the specimen, even though the failure could not visually be seen from the outside.

Maximum recorded force was 251.3 kN, while lowest was 203.7, resulting in $R_{failure}$-ratios in the range of 1.30 to 1.60 with an average of 1.48. This indicates that the Eurocode underestimated the capacity by approximately 50 % for these specimens. Having said that, 4 samples do not provide enough data to draw strict conclusions. Failure occurred as splitting parallel to grain on one of the side columns for all samples, see figure 4.19a.

After failure, when the steel plates were to be released, the dowels on the failure side were very bent and difficult to remove, as shown in figure 4.19b. This was, on the other hand, not the case for the side where splitting did not occur.

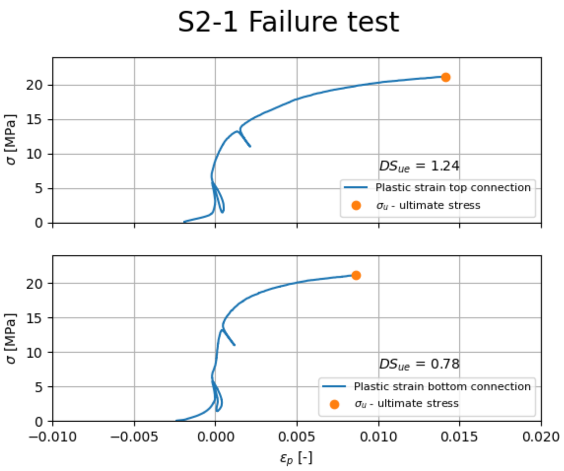


(a) Failure of S2-4 (b) Bent dowels after failure

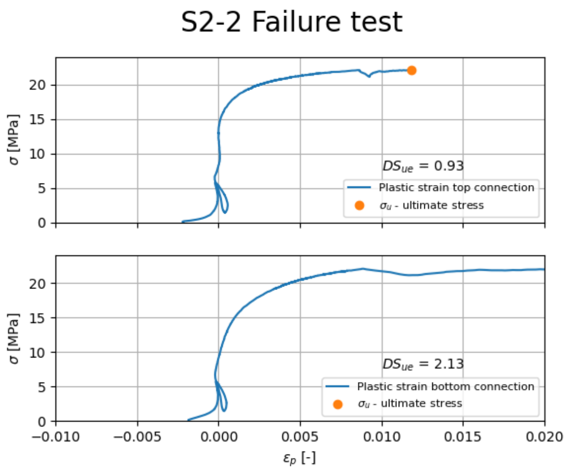
Figure 4.19: Failure of specimen type 2

Table 4.14: Failure values for specimen type 2

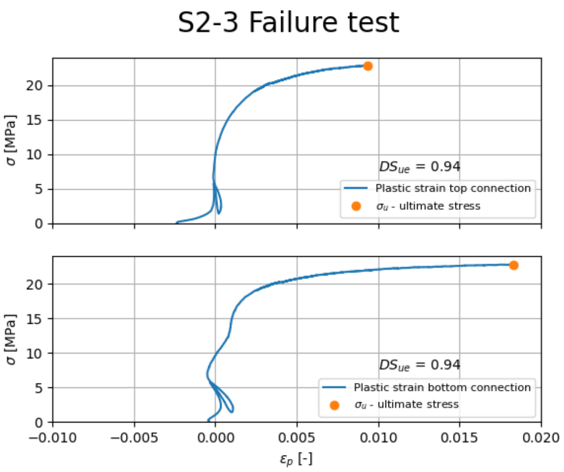
Specimen	Connection	$\Delta u_{failure}$ [mm]	$F_{failure}$ [kN]	$R_{failure}$ [-]	Ds_{ue} [-]	ϵ_i [-]
S2-1	Top	3.29	232.7	1.48	1.243	0.22581
	Bottom	2.65	232.7	1.48	0.778	0.28362
S2-2	Top	3.20	243.1	1.55	0.929	0.26174
	Bottom	4.50	243.1	1.55	2.132	0.22146
S2-3	Top	2.60	251.3	1.60	0.939	0.28225
	Bottom	4.57	251.3	1.60	0.940	0.05871
S2-4	Top	1.96	203.7	1.30	0.781	0.06806
	Bottom	3.55	203.7	1.30	1.529	0.21034



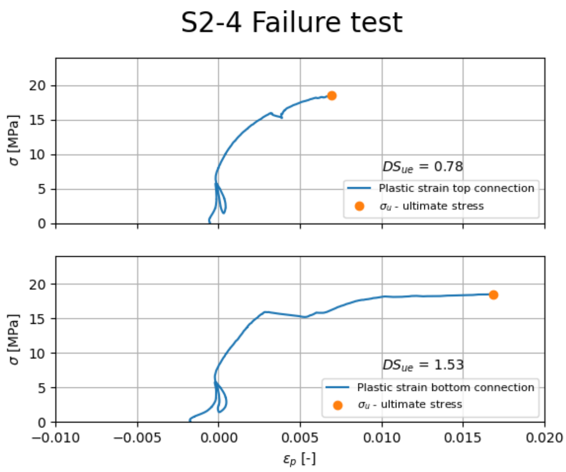
(a) S2-1



(b) S2-2



(c) S2-3



(d) S2-4

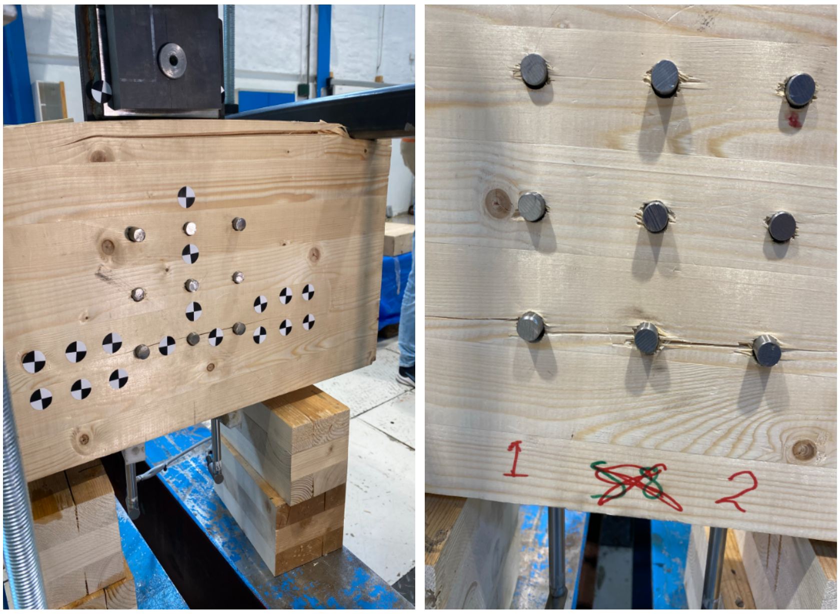
Figure 4.20: Failure curves S2

4.1.3.3 Failure | Specimen type 3

In figure 4.22, the failure curves for the three specimen type 3 samples are displayed. Failure values are presented in table 4.15. The failure curve for S3-2, figure 4.22b, indicates that slightly wrong E-modulus was obtained in the calculation, as embedment stress of 8 MPa needs to be reached before displaying plastic strain. This, on the other hand, was not the case for S3-1 and S3-3, displaying reasonable stress-strain relation at stress right above 2 MPa.

Maximum obtained force was 168.4 kN and minimum 141.4 kN. $R_{failure}$-values are within 1.9 and 2.3 with 2.17 in average. The ductility is found to be quite similar for S3-1 and S3-3. Due to the possibly wrong Young's modulus in S3-2, this value deviates a lot from the other two values.

Splitting perpendicular to grain was the predicted failure mode, and occurred to all samples, but the hold down of the specimen did probably not provide pure shear stress in the connection, which is the basis of the capacity calculation in Eurocode 5, see equation (2.9). The support conditions resulted in compression forces in the top of the specimens, which might have influenced crack propagation and maximum applied force. At least this should be kept in mind when using the results. Some cracks occurred due to compression forces from the supports. This issue could have been neglected if the specimens were wider, allowing the supports to be in a fair distance away from the connection. Photos of the supports and failure mode is shown in figure 4.21.

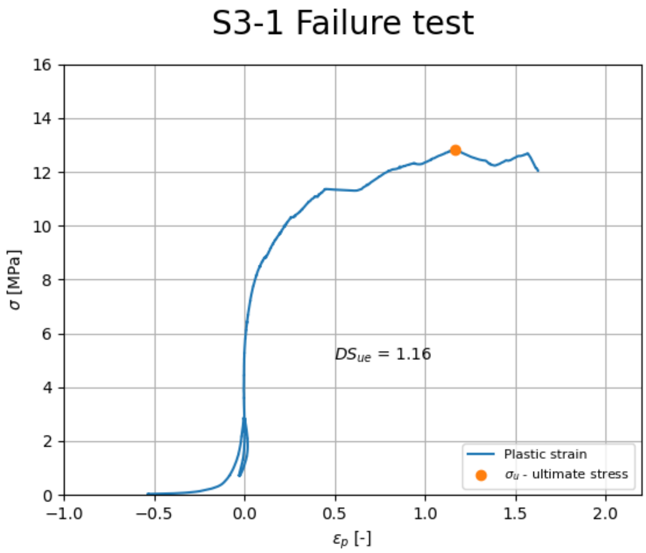


(a) Failure due to compression (b) Splitting ⊥ grain

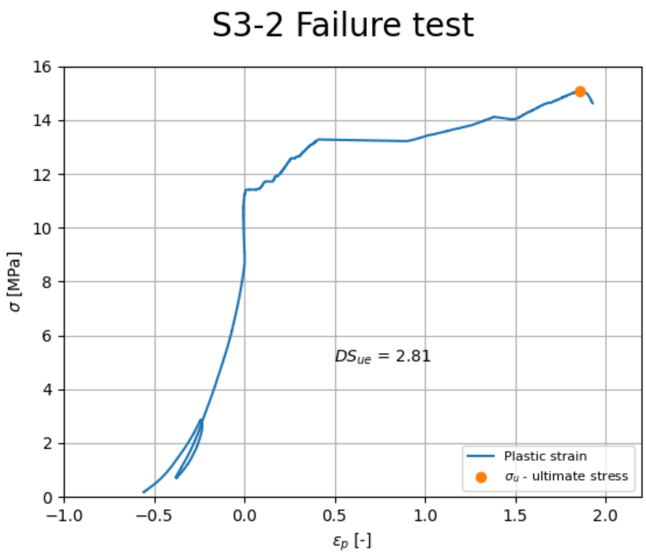
Figure 4.21: Failure of specimen type 3

Table 4.15: Failure values for specimen type 3

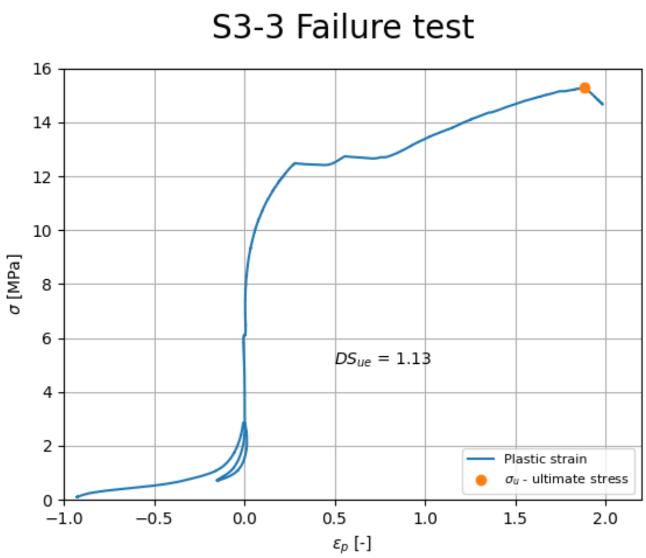
Specimen	Connection	$\Delta u_{failure}$ [mm]	$F_{failure}$ [kN]	$R_{failure}$ [-]	Ds_{ue} [-]	ϵ_i [-]
S3-1	Bottom	2.7	141.4	1.9	1.16	0.0048
S3-2	Bottom	3.1	168.4	2.3	2.81	0.0051
S3-1	Bottom	4.5	168.4	2.3	1.13	0.0084



(a) S3-1



(b) S3-2



(c) S3-3

Figure 4.22: Failure curves S3

4.1.3.4 Failure | Comparison and remarks

The following interesting observations have been done:

- In general, the Eurocode underestimates the capacity in the tested connections. This conclusion is drawn on the basis of all $R_{failure}$-values being above 1, varying from 1.20 to 2.30. It seems like the Eurocode underestimates specimens loaded perpendicular to grain even more than parallel to grain, although the amount of data analyzed in the present thesis is not enough to draw any strict conclusions as the estimated capacities were based on mean values. Consequently, deviations in material properties cannot be disregarded.

- For most cases, the ductility for the connection that ran till failure was distinctively lower than the opposite.

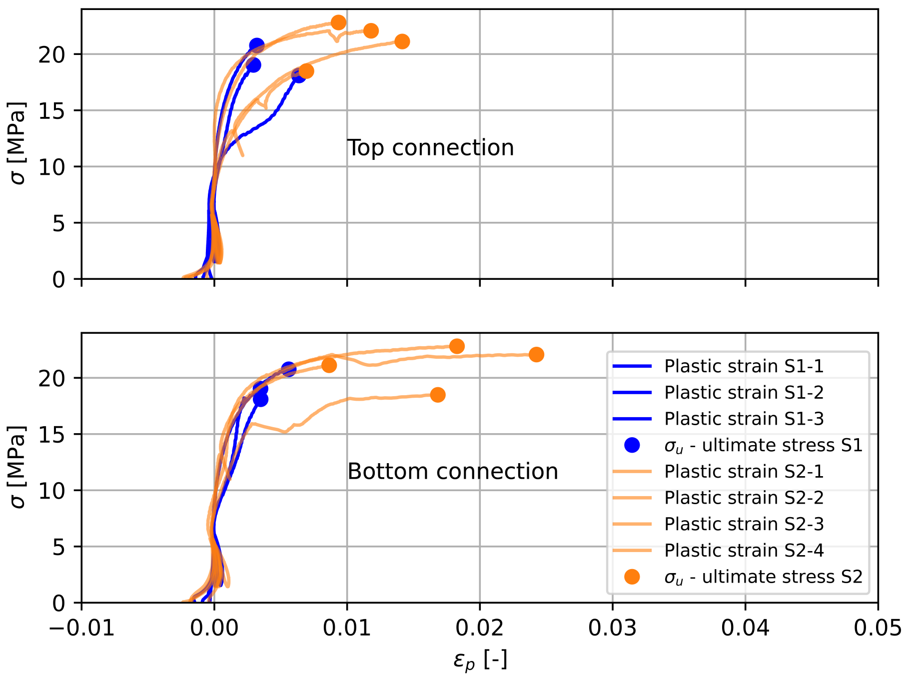


Figure 4.23: Comparison of failure test for specimen type 1 and 2.

- Especially for specimen type 2, yielding of the steel dowels ensured a ductile failure as can be seen in figure 4.23. This is reflected through higher ductility-values, Ds_{ue}, compared to specimen type 1, where no such clear bending of the dowels was possible to observe. The ductility is found to be slightly higher for specimen type 3, but it is important to remember that the connection was not exposed to pure shear only, as discussed in section 4.1.3.3

- Figure 4.24 shows a comparison of the calculated slip-strain for the three different configurations. The tendency is that the slip-strain increases for an increased number of fasteners parallel to grain. For loading perpendicular to grain with three dowels in three rows, the calculated slip-strain is a lot higher than for the same number of fasteners in grain direction. This may be a result of the oversized holes as discussed in 3.1.4.

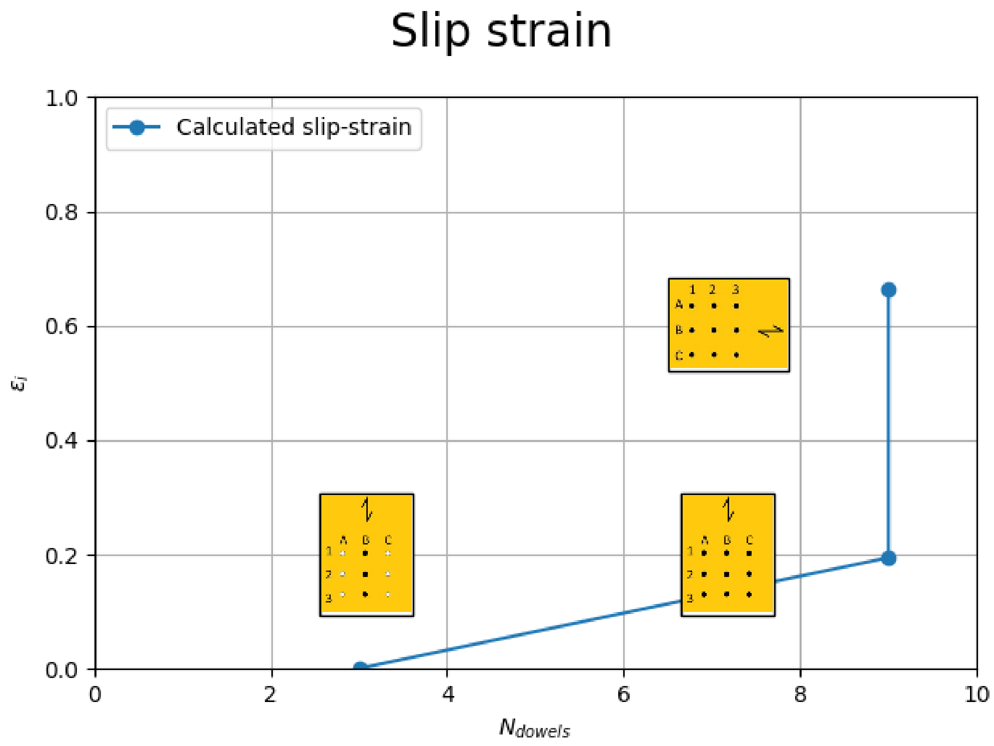


Figure 4.24: Comparison of calculated slip strain for specimen type 1, 2 and 3.

- One should keep in mind that there are several ways to calculate ductility. Only one method is presented here, meaning that it is important to know which parameters that are included when comparing the calculated ductilities here to other experimental data.
- For all measurements for S1 and S2, it was challenging to pick a stiffness that was correct for both top and bottom connection. For that reason, all curves do not display perfect plastic strains, such as figure 4.18c and figure 4.20c. For the last mentioned, the top and bottom connection get the same ductility value, even though one can clearly observe that the bottom connection behaves more ductile. This may be caused by the bottom connection having approximately half the calculated Young's modulus as the top connection and thus getting a lower ductility value per equation 2.20.

4.2 Numerical results

The results from the analysis performed in the FE-model is presented in the following sections. All obtained numerical results are compared to the experimental results in order to illustrate the accuracy of the numerical model.

4.2.1 Mesh convergence study

Figure 4.25 shows how the model stiffness changed for varied mesh sizes of the different parts of the model. As can be seen, the dowel, ring 1 and ring 2 are the most sensitive to change in mesh size. As these parts are connected by tie-constraints, some issues may arise from the master-surface having a larger mesh size than the slave-surface in the mesh study. The mesh has therefore been chosen so that element size increases from dowel to ring 2. It is also worth noting that the dowel and ring 1 are partitioned in a way that prohibits large elements in these parts. The study shows that the stiffness is not influenced by the change to a coarse mesh in the outer timber part or the steel plates. Here the mesh that gets the best combination of calculation time and accuracy is chosen.

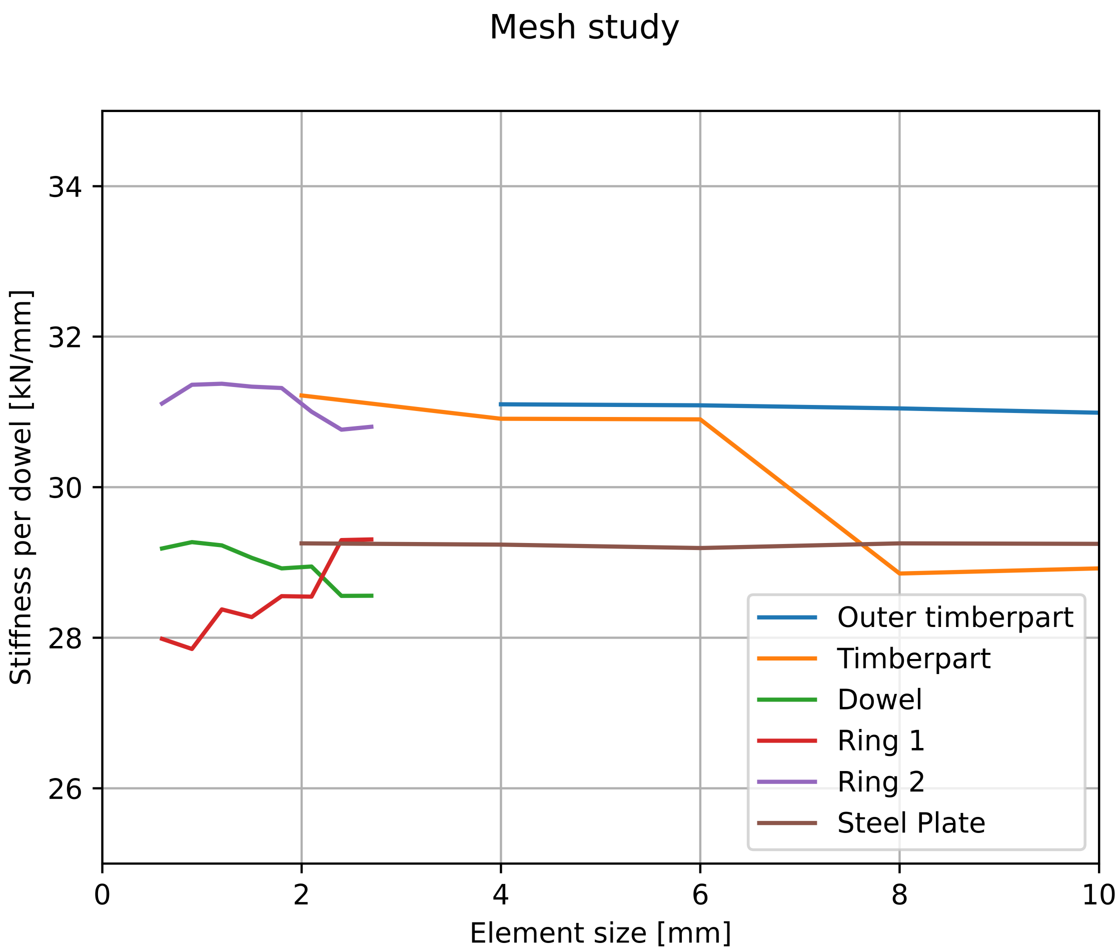


Figure 4.25: Mesh convergence for specimen S1 with 1 dowel and 180 degree contact angle

Based on the results of the mesh study two different meshes were made available in the Python script. A coarse mesh, used for stiffness optimization and angle iterations, and a fine mesh used in the failure test simulation. The chosen mesh size for the different parts of the model is presented in table 4.16 below.

Table 4.16: Results from mesh convergence study on S1 with 3 rows of dowels

Part	**Mesh**	**Element size [mm]**	**Elements per part**
Dowel	Fine	0.6	61 440
	Coarse	1.8	2 560
Ring 1	Fine	0.8	10 752
	Coarse	2	624
Ring 2	Fine	1	9 180
	Coarse	2.2	644
Timber part	Fine	3	30 440
	Coarse	5	7 666
Outer timber part	Fine/Coarse	10	1 353
Steel plate	Fine/Coarse	4	5 374

The resulting number of elements and calculation time for the different specimens are shown in 4.17. As can be seen, iteration based on fine mesh is not suitable due to extensive calculation time.

Table 4.17: Calculation time for the different setups with different mesh size

Test specimen	**Mesh size**	**Total number of elements**	**Total number of variables**	**CPU time [s]**
S1-B123	Fine	342 047	890 067	1 850.0
	Coarse	39 545	89 856	81.3
S2-A123B123C123	Fine	1 063 469	2 642 997	7 535.8
	Coarse	125 192	225 741	250.4
S3-A123B123C123	Fine	1 079 844	2 667 483	8 980.2
	Coarse	131 541	245 394	445.7

4.2.2 Stiffness

In order to optimize the numerical model in such a way that it returned equal stiffness as the experimental results from the lab, three main parameters were investigated. An overview of the parameters and a description is given in figure 4.26. Note that also other parameters could have been varied, as the numerical model is fully parametric. Due to limited time available, a certain amount of parameters had to be chosen for the present thesis.

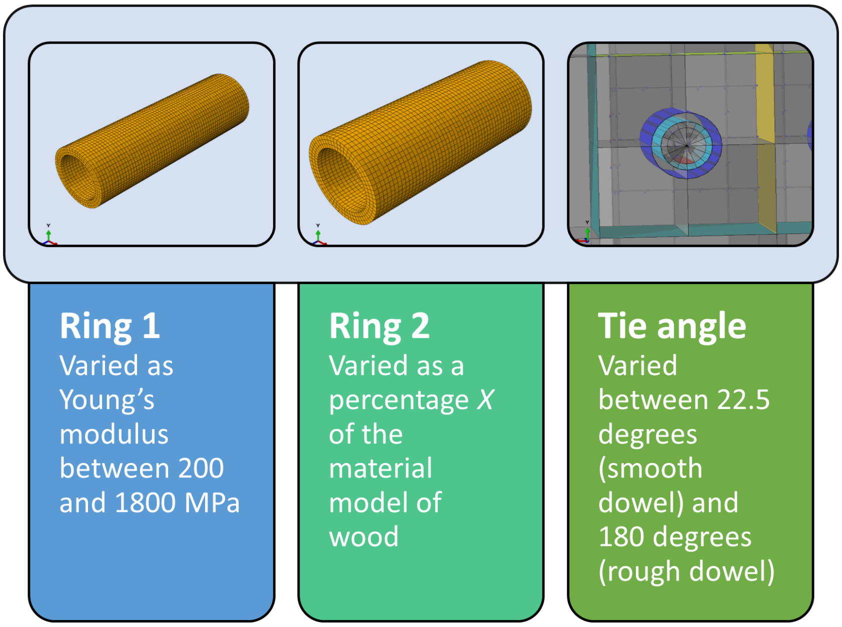


Figure 4.26: Work flow investigating stiffness in numerical model

Different configurations were ran in the numerical model to investigate how the parameters were affected. The same notation system as introduced in section 3.1.5.1 (figure 1) was utilized. As cyclic tests on different dowel configurations were not performed in the present thesis, the results obtained by Frette et al. (2021) for the respective configurations were used.

4.2.2.1 Stiffness | Parameter study of ring 1 and 2

Figure 4.27 shows the results from the parameter study for ring 1 and 2 of configuration S1-B123 (i.e. specimen type 1 with three dowels in grain direction). Similar plots were made for all configurations, and may be seen in appendix D. This particular plot is, however, included in order to illustrate how the parameters influence each other. The corresponding Young's modulus in ring 1 and 2, denoted E_{r1} and E_{r2}, needed to obtain the measured stiffness in the different configurations are displayed in table 4.18, table 4.19 and table 4.20 for specimen type 1, 2 and 3 respectively. Ring 2 was iterated for three different values; 0.5, 1.0 and 1.5 times the material model of wood, E_{wood}. The iteration of ring 2 may be seen as the three different graphs in figure 4.27. Here, any number could have been chosen, but 0.5, 1.0 and 1.5 was the decision made by the authors for the present thesis in order to delimit the number of simulations. The iteration of E_{r1} is represented on the x-axis, while the corresponding stiffness is displayed on the y-axis. The red X-marks represent the necessary Young's modulus of ring 1 in order to obtain the measured stiffness, and these values are tabulated in the tables mentioned above.

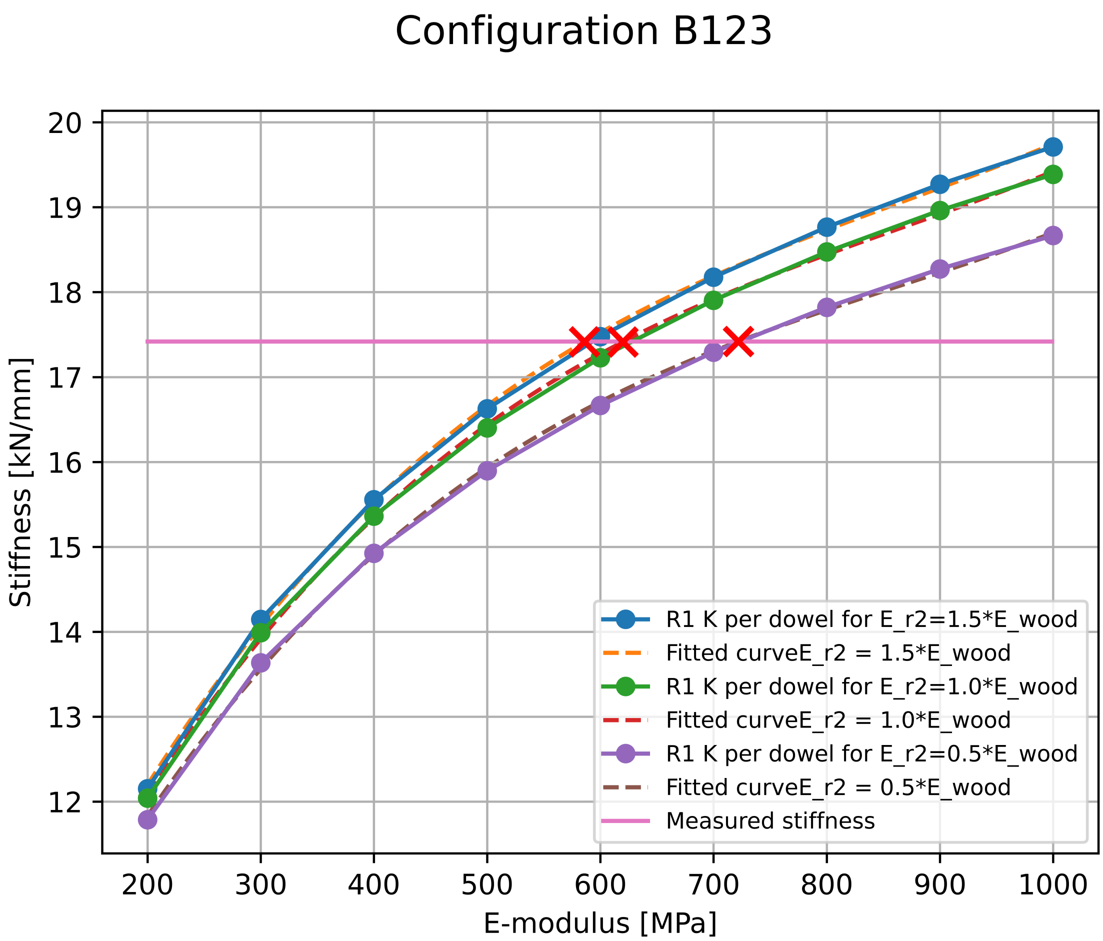


Figure 4.27: S1-B123

Table 4.18: Results from numerical analysis of specimen type 1

Configuration	E_{r2} $[\mathrm{X} \cdot \mathrm{E_{wood}}]$	E_{r1} [MPa]	$K_{measured}$ [kN/mm]
S1-B1	0.50	2452	28.77
	1.00	2845	
	1.50	4222	
S1-B12	0.50	462	17.29
	1.00	482	
	1.50	539	
S1-B123	0.50	586	17.42
	1.00	620	
	1.50	722	

Table 4.19: Results from numerical analysis of specimen type 2

Configuration	$\boldsymbol{E_{r2}}$ **[X · E$_{\textbf{wood}}$]**	$\boldsymbol{E_{r1}}$ **[MPa]**	$\boldsymbol{K_{measured}}$ **[kN/mm]**
S2-B1	0.50	901	24.64
	1.00	969	
	1.50	1182	
S2-B12	0.50	267	14.71
	1.00	274	
	1.50	294	
S2-B123	0.50	220	12.77
	1.00	224	
	1.50	236	
S2-A1B1C1	0.50	572	19.04
	1.00	605	
	1.50	703	
S2-A12B12C12	0.50	378	14.19
	1.00	393	
	1.50	432	
S2-A123B123C123	0.50	318	12.12
	1.00	330	
	1.20	360	

Table 4.20: Results from numerical analysis of specimen type 3

Configuration	$\boldsymbol{E_{r2}}$ **[X · E$_{\textbf{wood}}$]**	$\boldsymbol{E_{r1}}$ **[MPa]**	$\boldsymbol{K_{measured}}$ **[kN/mm]**
S3-B1	0.50	185	10.51
	1.00	202	
	1.50	246	
S3-B123	0.50	43	5.27
	1.00	44	
	1.50	48	
S3-A2B2C2	0.50	169	9.26
	1.00	184	
	1.50	226	
S3-A123B123C123	0.50	63	6.02
	1.00	66	
	1.50	71	

As can be seen in table 4.18 to table 4.20 and figure 4.27, ring 2 tends to have less influence than ring 1. In order to tune the numerical model such that it can be used for any geometry, it was investigated how E_{r1} varied as a function of rows with fasteners, keeping E_{r2} constant equal to $1.0 \cdot E_{wood}$.

In figure 4.28, all data for 1, 2 and 3 rows of fasteners are gathered for specimen type 1 and 2, i.e. force applied parallel to grain. Figure 4.29 shows the same, though for force perpendicular to grain (specimen type 3). The trend line is the fitted line approximating the data from the numerical analysis. Table 4.21 presents the final suggested E_{r1}-value for 1, 2 and 3 rows of fasteners respectively. One should keep in mind that this estimation does not take number of fasteners in the same row into account, meaning that the model will return the same E_{r1}-value for a configuration with, for instance, 1 and 3 fasteners in the same row.

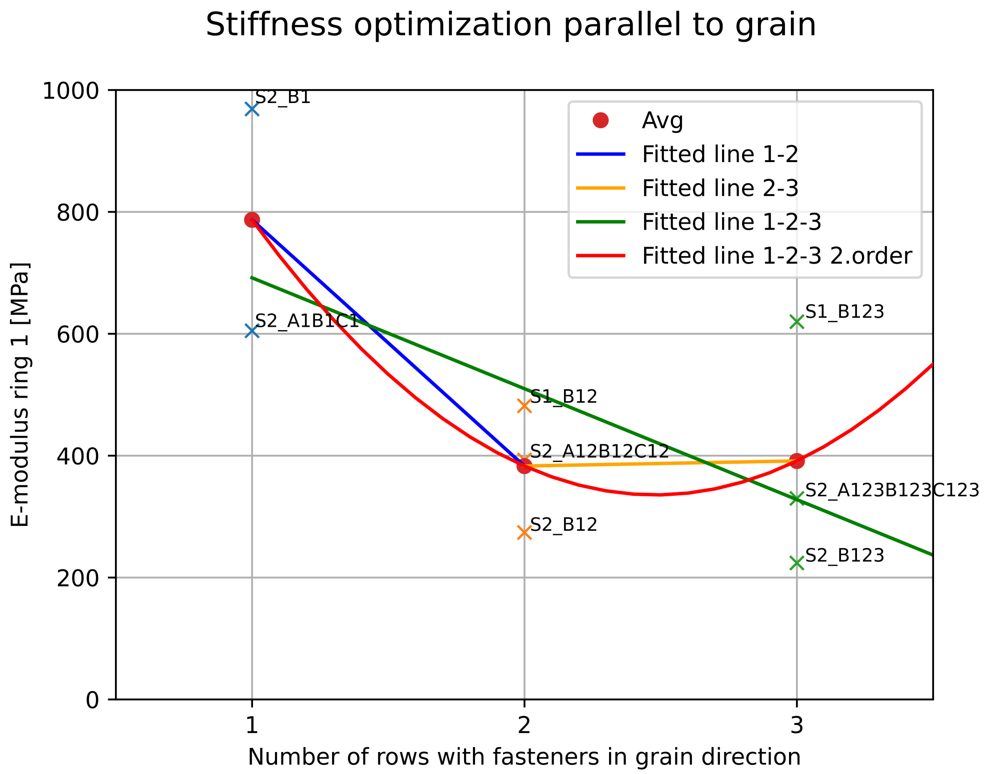


Figure 4.28: Ring 1 stiffness, E_{r1}, for rows of fasteners parallel to grain

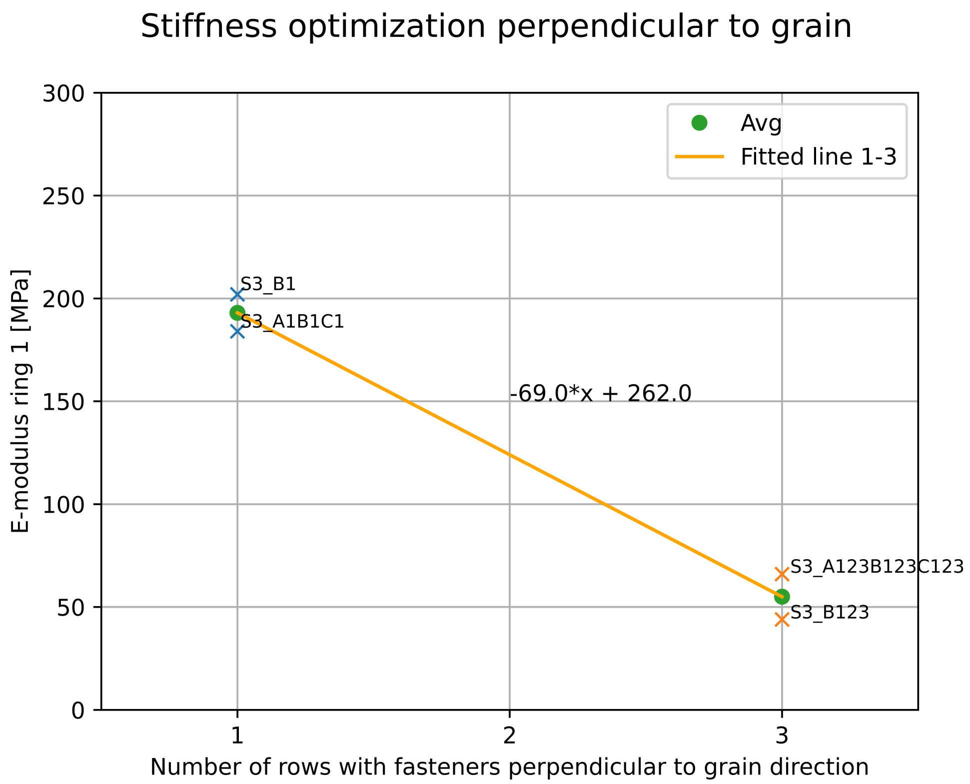


Figure 4.29: Ring 1 stiffness, E_{r1}, for rows of fasteners perpendicular to grain

Table 4.21: Estimated E_{r1}-values from figure 4.28 and figure 4.29

Number of rows with fasteners	$\boldsymbol{E_{r1,parallel}}$ **[MPa]**	$\boldsymbol{E_{r1,perpendicular}}$ **[MPa]**
1	787.0	193.0
2	383.0	-
3	391.3	55.0

The following remarks should be kept in mind:

- Whether or not it is sufficient to only have number of fasteners in load direction as input to decide the stiffness of ring 1 is yet unclear based on the amount of data used in the present thesis. There is, however, a clear tendency that number of rows have a larger influence than number of fasteners in each row. This statement is based on results found in the present study, and verified by Frette et al. (2021), who concluded that configurations with three dowels in one row (e.g. A1B1C1) gave higher stiffness than three dowels in load direction (e.g. B123).

- When the fasteners are organized in several rows, the Eurocode reduces the contribution by an effective number of fasteners, n_{ef}, see equation (2.4). The same tendency is displayed in these results, showing that the stiffness in ring 1 needs to be smaller for fasteners in several rows compared to fasteners in the same row.

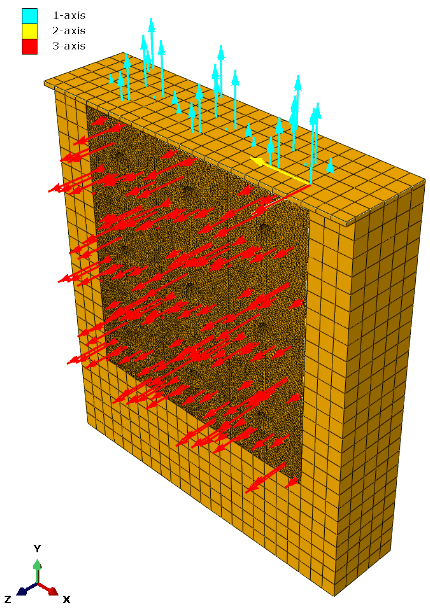
- The influence of stiffness in ring 2 seems to be greater for high values of E_{r1}, implying that the assumption of $E_{r2} = 1.0 \cdot E_{wood}$ might not be right for connections displaying high stiffness.

- A weakness of tuning the numerical model based on experimental data from both S1 and S2, is that specimen type 1 and 2 were of different strength class; T15 and T22 respectively. For similar configurations, somewhat different experimental stiffnesses were achieved which, according to Frette et al. (2021) was caused by different tensile strength in the different specimens. Nevertheless, the differences are not considered to be causing huge errors in the presented results.

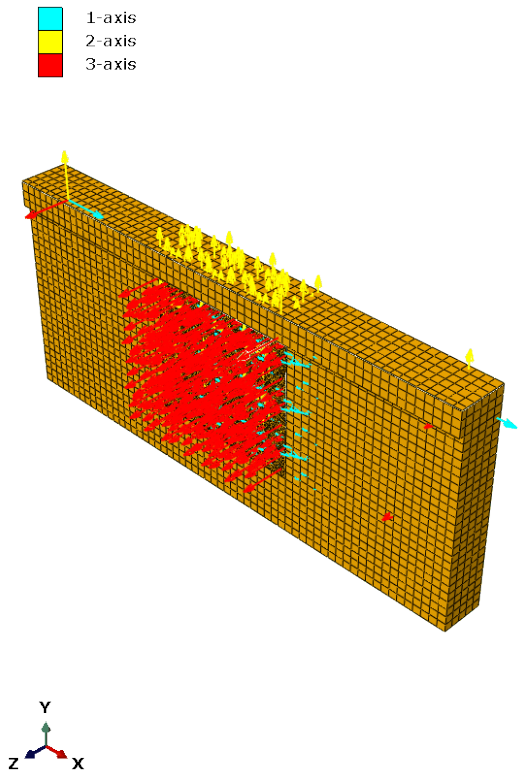
- The following three different trend lines are included for figure 4.28. These three fitted lines are included to illustrate why it is difficult to draw any final conclusions based on the present data. For the limited data available, it seems most reasonable to use fitted line number 1 from the list below, while 2 and 3 may be useful when more data is analyzed, see also section 5.3.

    1. Fitted line of 1.order between the calculated average of 1 and 2 number of rows with fasteners in grain directions, and 2 and 3.
    2. Fitted line of 1.order between 1, 2 and 3 number of rows with fasteners in grain direction.
    3. Fitted line of 2.order between 1, 2 and 3 number of rows with fasteners in grain direction.

- Due to lack of available data, only four different configurations with 1 and 3 rows with fasteners respectively, were ran for specimen type 3. Consequently, a fitted line of 1.order between the averaged values is the only possible estimation.



(a) Material orientation S1 and S2



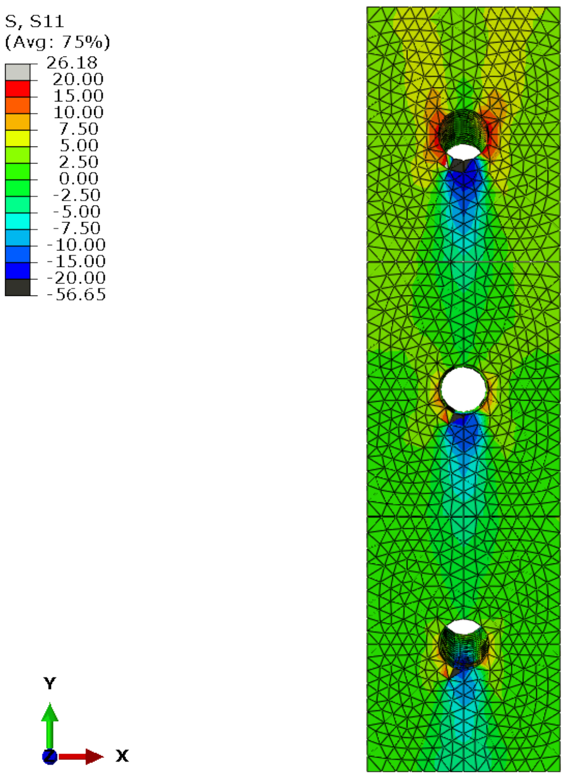
(b) Material orientation S3

Figure 4.30: Material orientation Abaqus

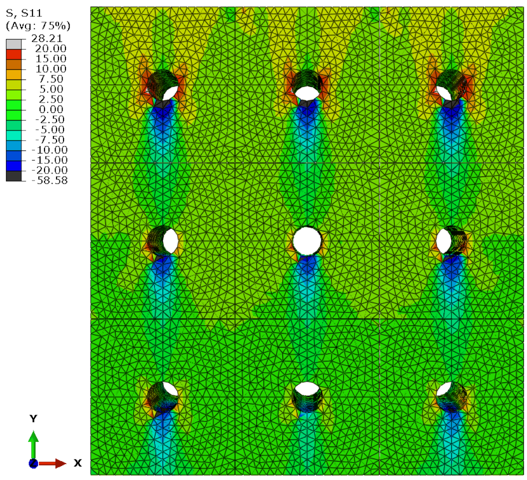
The remarks commented above may be further confirmed by investigating the stress-field distribution from the Abaqus Output Database. The material orientation, which is important to establish in order to understand the color plots, is displayed in figure 4.30. The stress distribution for specimen type 1, 2 and 3 is displayed in figure 4.31. S11 stress is shown for specimen type 1 and 2, while S22 applies for type 3.

These interesting observations have been made from the stress distribution plots:

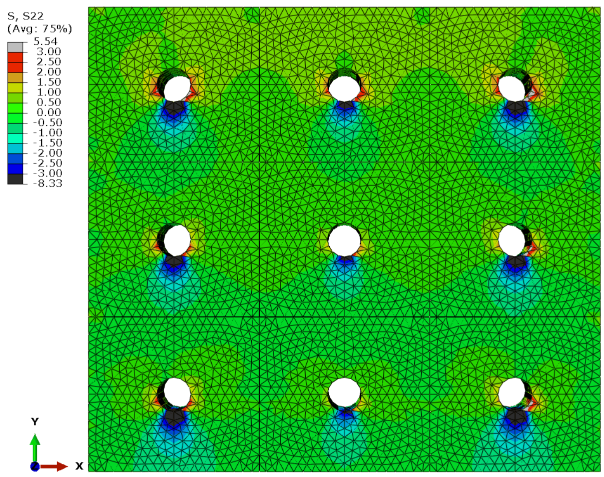
- The row of dowels closest to the applied load displays highest stress. This verifies the reduction factor n_{ef}, effective number of fasteners in several rows, and that the stiffness in ring 1 should be reduced for an increased number of rows as seen in figure 4.28 and figure 4.29.
- Reasonable stress is obtained in the numerical model, with some exceptions of numerical noise. The obtained stress is within the elastic stress range.
- For S2 lower stresses develop in the middle than in the outer parts. Corresponding well with the failure seen in the outermost part in the failure tests in the lab.



(a) S11 stress in specimen S1 at F = 19.6 kN



(b) S11 stress in specimen S2 at F = 56.0 kN



(c) S22 stress in specimen S3 at F = 31.2 kN

Figure 4.31: Developed stresses in load direction for specimen type 1,2 and 3 in inner timber part

4.2.2.2 Stiffness | Parameter study of tie angle

The tie angle between the inner ring and dowel was varied from 22.5°and 180°. The result is shown in figure 4.32. This analysis was carried out for specimen type 1 only, but it is believed with a high level of certainty that the same qualitative results would have been displayed for specimen type 2 and 3 as well. The step between each analysis was 22.5°and it is evident from the results that the stiffness increases with increased tie angle, where the increase is greater for low than for high tie angles.

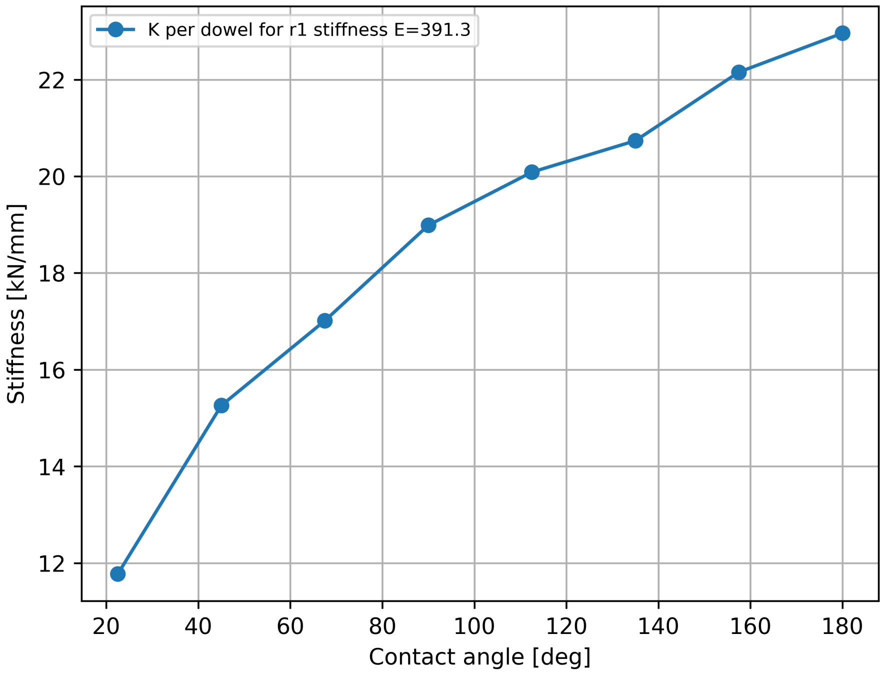


Figure 4.32: Angle of tie connection in degrees

As described in section 2.7, Dorn (2012) found the smoothness of the dowel to be important regarding the contact angle between the dowel and timber. Tests with smallest tie angle, i.e. 22.5°, displayed a stiffness right below 12 kN/mm, while the 180°-tie angle displayed nearly twice as large stiffness. This shows the importance of choosing an appropriate tie angle when investigating stiffness in numerical models.

An extensive parameter study of tie angle was not carried out in the present thesis due to limited time available. The goal was rather to verify the findings of Dorn (2012) and to show that tie angle influences the numerical model. A tie angle of 45°was chosen by the authors for all analyses presented in section 4.2.2.1 and section 4.2.3 corresponding to a smooth dowel.

4.2.3 Failure test

Results from the failure test are displayed in figure 4.33 to figure 4.35 for specimen type 1 to 3 respectively. The mesh was varied between fine and coarse with either 0 or 2 rings. All experimental results are displayed in the same graph. These simulations were conducted with a constant stiffness of $E_{r1} = 500$ MPa only, and due to time limitations, analysis with optimized stiffnesses were not performed.

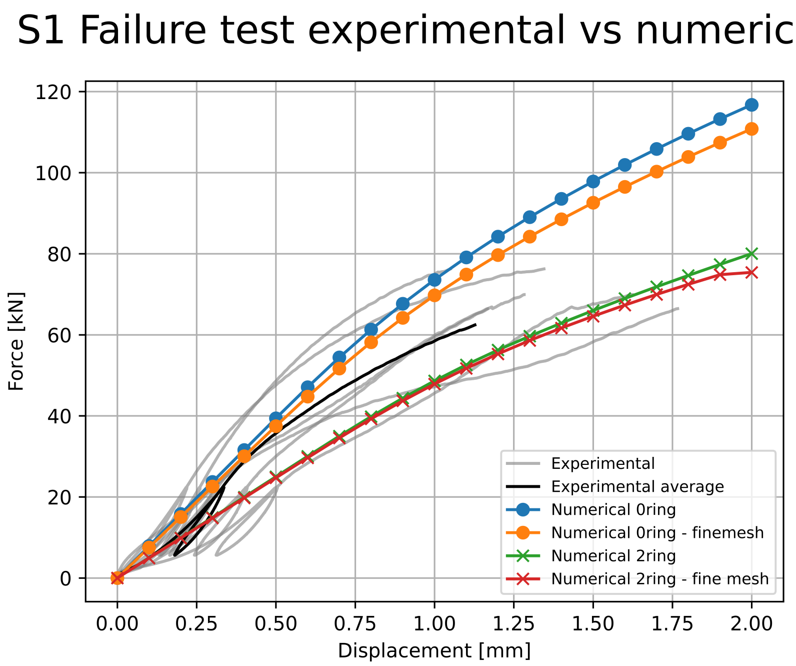


Figure 4.33: Experimental data compared to numerical model specimen S1

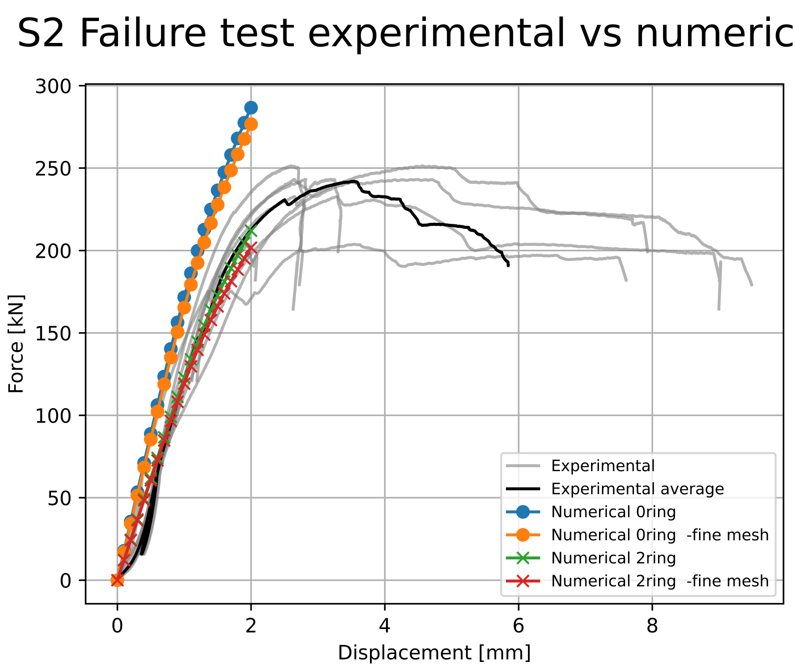


Figure 4.34: Experimental data compared to numerical model specimen S2

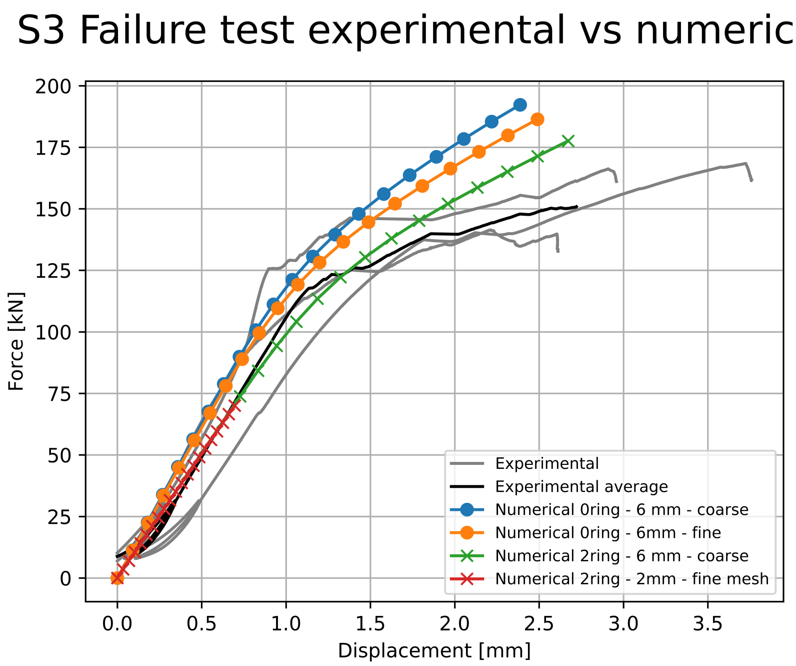


Figure 4.35: Experimental data compared to numerical model specimen S3

The following observations and remarks are made on the basis of the obtained results:

- In general, the numerical results coincide well with the experimental results, especially for the linear-elastic part. This shows that the ductile failure mode with plastic hinges in the dowels are well achieved in the numerical model.

- The reason for non-perfect correspondence in the non-linear parts, i.e. at the beginning and end of the failure test, is due to the plastic material model in the steel part only. In order to achieve more accurate results, a plastic material model may be implemented for the wood as well. However, it is evident from the results that the steel dowels give an important contribution to the failure mode.

- The difference between 0 and 2 rings is greater than the difference between fine and coarse mesh. For all instances, tests with 0 rings are found to be too stiff. The largest deviation between 0 and 2 rings were found for specimen type 2 at 2 mm displacement, where 40 % higher force were recorded for tests with 0 rings at most. Fine mesh gives a slightly less stiff model compared to coarse mesh both for 0 and 2 rings as expected, the deviation increases for larger displacements in the model.

- The largest deviation between the experimental and numerical results are found close to material failure. The reason for this is that no failure criteria is implemented in the numerical model.

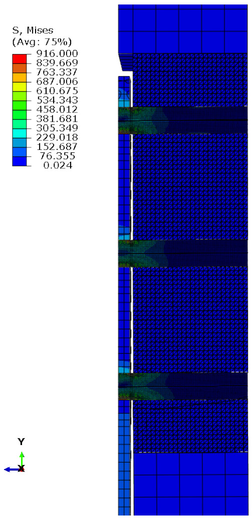
- One should keep in mind that the presented results are only compared with experimental data produced in the present thesis. A completely finished numerical model should be able to provide reliable data for any geometry. To achieve this, further tuning of the numerical model is required, see also section 5.3. The experimental data is still included in order to qualitatively give an impression of the accuracy of the model.

- For S3, the specimen had to be given a larger displacement of 6 mm in the analysis to account for bending of the specimen. The plotted displacement is the relative displacement of the steel plate compared to the top of the outer timber part, like the measurement with LVDTs in the lab confer figure 3.7c. The flat part of the failure curves from lab were also removed to neglect the effect of the oversized holes as discussed in section 3.1.4.

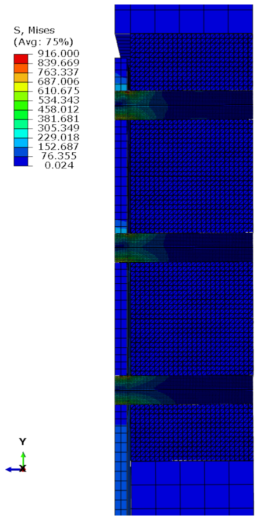
In addition to the presented failure curves, some interesting and important observations may be seen in the stress distribution from the Abaqus output file. For this, only the results from specimen type 2 are included in the thesis. The results for specimen type 1 and 3 shows qualitatively the same.

Observations:

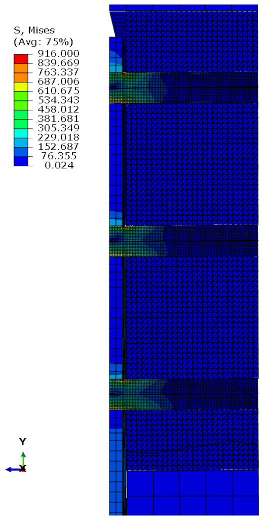
- It can be seen that the stress is slightly higher for the outer dowels (column 1 and 3 according to figure 3.8a). This corresponds very well with the displayed failure mode for specimen type 2, where splitting occurred parallel to grain for the outer columns of dowels.
- The ductile failure mode, forming plastic hinges in the dowels, are displayed in the numerical model. First yielding of the steel dowel occurs at 0.5 mm, but plastic hinges are not clearly visible before 1 mm.
- The steel plate develops relatively high stresses at 2 mm displacement when all the load is transferred from the timber to the steel.
- The rotation seen in the lower timber part away from the symmetry plane, corresponds well with the observed failure in the lab tests.



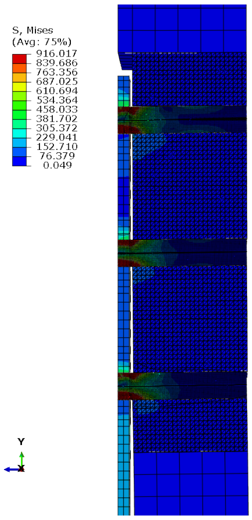
(a) Column 1 - 0.5 mm



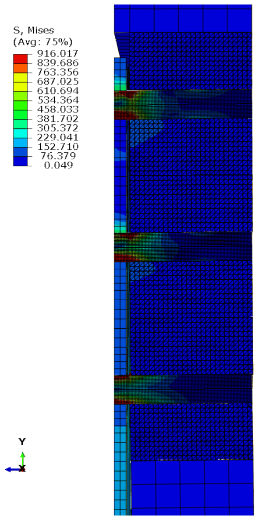
(b) Column 2 - 0.5 mm



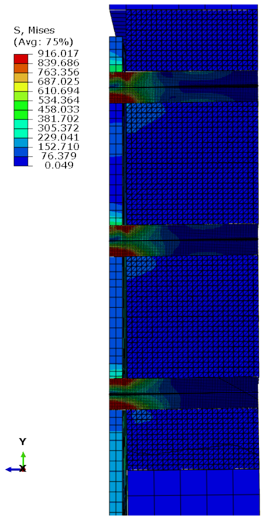
(c) Column 3 - 0.5 mm



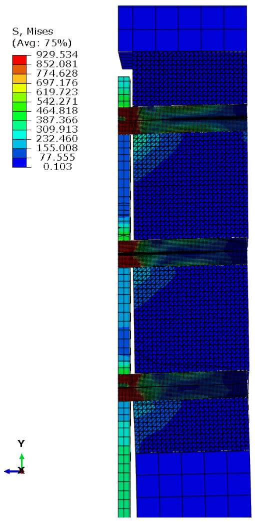
(d) Column 1 - 1.0 mm



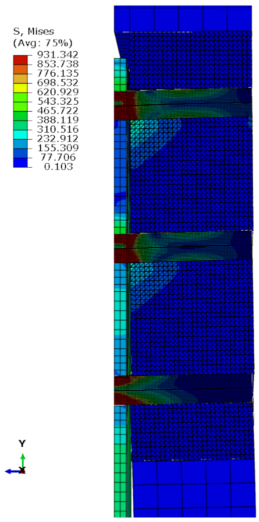
(e) Column 2 - 1.0 mm



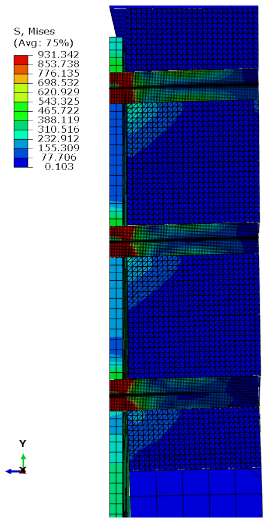
(f) Column 3 - 1.0 mm



(g) Column 1 - 2.0 mm



(h) Column 2 - 2.0 mm



(i) Column 3 - 2.0 mm

Figure 4.36: Development of von Mises stresses in the steelparts in column 1,2 and 3 in specimen S2

4.2.4 Experimental use of FE-model

In the present thesis, numerical results have been compared to measured stiffnesses from lab tests. As the FE-model is fully parametric this section is meant to show some results that cannot be directly compared to lab tests in the current thesis, but that may be explored further to validate the FE-model. In figure 4.37, the trending stiffness per dowel per shear plane for a specimen of the same cross section as S1, but with different length and number of dowels is shown. In the tests, both Young's modulus of ring 1 and ring 2 is held constant. Implying that the only influence giving the lower stiffness, is the addition of more fasteners. The loading is set to be 40% of F_{est} for the given dowel configuration.

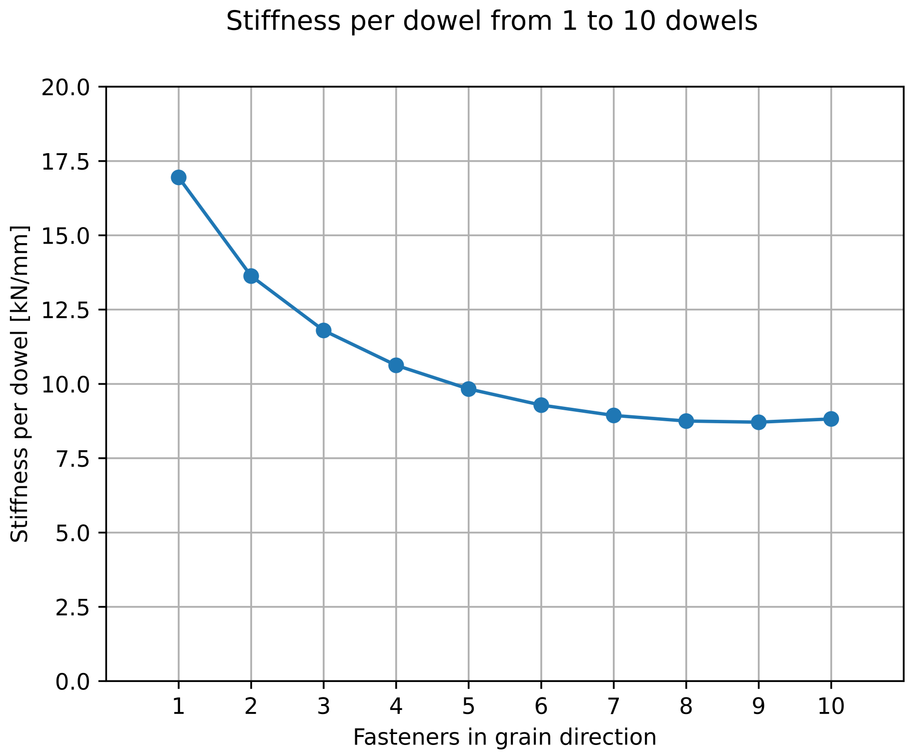
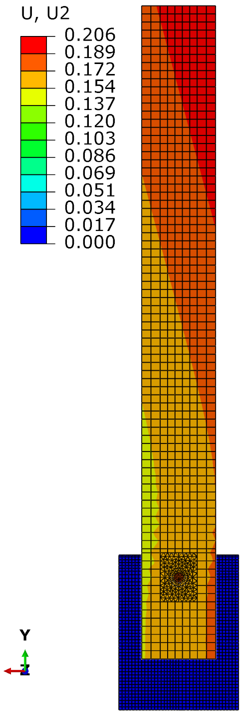


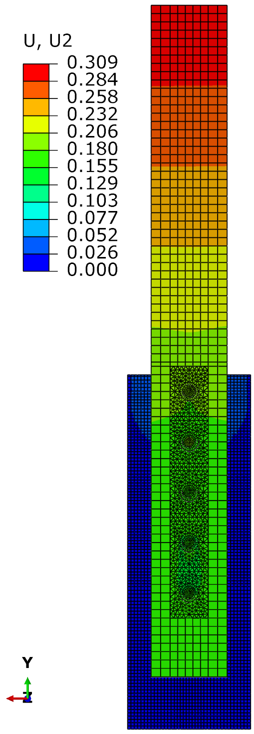
Figure 4.37: Stiffness trend with increased number of fasteners in fiber direction

Observations:

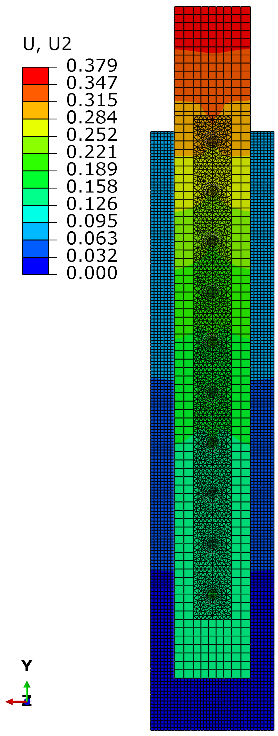
- The stiffness trends lower for an increased number of fasteners in load direction, but seems to reach a lower boundary at 7 fasteners stabilizing around 8.8 kN/mm.
- From lab testing, only results up to 3 fasteners in load direction were available. As figure 4.37 show, there is a steep decline in stiffness per dowel from 1 to 3 fasteners before the stiffness seems to level out. It would have been interesting to investigate if a similar, lower boundary trend could be found in lab tests.
- These results must be interpreted with caution. As shown in figure 4.38, there were some rotation in the test with 1 fastener. It is unknown to the authors what the origin of this rotation is. Also as can be seen by the displacement plot, the model get quite a lot of deformation from the wood itself, since displacement is measured at the top of the timber part this can affect the calculated stiffness.



(a) 1 fastener



(b) 5 fasteners



(c) 10 fasteners

Figure 4.38: How the model looked for 1,5 and 10 fasteners in load direction

Chapter 5

Concluding remarks

This chapter presents some concluding remarks found through the master thesis. Finally, potential sources of error and some suggestions for further work are presented.

5.1 Conclusions

The scope of this thesis was to gather data to be used for numerical modelling of dowel-type connections and to investigate the Eurocode-formula. The main concluding remarks are presented below:

- The Eurocode-calculation of stiffness is found to be too simplified. The Eurocode tends to underestimate the actual stiffness in dowel-type connections. The level of underestimation seems to be higher for loading parallel to grain than perpendicular. This implies that a coefficient taking angle-to-grain into account might have been included in the formula.
- Another parameter that could have been included is a coefficient taking load level or utilization of the connection into account. The results presented in the present thesis shows, without exceptions, that the connection stiffness increases for increased utilization, meaning that the deviation between the estimated and actual stiffness becomes larger for highly utilized connections. At most, an actual stiffness 2.6 times higher than the Eurocode value was recorded. Given the importance of designing efficient structural systems, such a coefficient might be useful for designers. Still it is important to remember that the Eurocode provides a framework of rules having safety as the primary factor. Therefore, the Eurocode seems reasonable as long as the actual stiffness is higher than the calculated.
- For the numerical modelling, the approach with two rings close to the dowel tends to be necessary in order to generate results that are not too stiff. The inner ring seems to have a larger influence than the outer ring, but further investigations and tuning should be carried out before a final conclusion can be drawn here.
- As the results from the experimental work presents, increased loading gives a higher connection stiffness. The stress plots from the FE-analysis show a non-uniform distribution of stresses per dowel, some dowels get higher stresses than others. This can explain the reduced stiffness measured by Frette et al. (2021) for configurations with multiple fasteners in load direction.

5.2 Sources of error

As discussed throughout the thesis, various sources of error influenced the results in some way or another. The most important errors are listed below:

- The placement of LVDTs not in a fully upright position. Small deviations for the LVDTs out of position could lead to errors in the measurements, influencing the calculated stiffness and viscous damping ratio. The LVDTs were adjusted through visual control, meaning human error cannot be excluded.
- Uncertainties regarding moisture content in the specimens as described in section 4.1.1.4. The MC was measured for each specimen to keep control of it, and the results showed quite constant MC, though with some small variations.
- The global measurement may be a bit imprecise, as it was challenging to keep the steel rods straight along the component.
- Errors in the data processing may not be disregarded, but the results obtained in the present thesis showed in general good correspondence to the results by Frette et al. (2021).

5.3 Further work

The following aspects would have been interesting to investigate further:

- Tuning of the numerical model. It is yet unclear whether it is sufficient to estimate the Young's modulus in ring 1 based on number of rows with fasteners only (see figures 4.28 and 4.29. To investigate this, testing with specimens with more than 3 rows of fasteners may be conducted.
- Large parametric study of the numerical model. As the model is fully parametric, different parameters may easily be changed. These parameters may, for instance, be varied:
  - Size of cross-section.
  - Thickness of steel-plates (Note that only one slotted-in steel plate may be investigated).
  - Diameter of the dowels.
  - Load amplitude in order to verify the findings from the experimental work.
  - Tie angle study to compare the results with experimental data.
- Failure criteria for wood. As for now, no failure criteria is implemented in the numerical model, meaning that the model will display wrong results for loading close to the estimated capacity. DIC recordings were conducted during the failure test, and these recordings may be useful to create strain fields and investigate crack propagation. The recordings are included in the digital appendix.
- Further development of the parametric numerical model, adding multiple plates and automatic load calculation.

Bibliography

Abrahamsen, Rune (2018). "Mjøstårnet - 18 storey timber building completed". en. In: p. 13.

Abrahamsen, Rune et al. (2020). "DYNAMIC RESPONSE OF TALL TIMBER BUILDINGS UNDER SERVICE LOAD – THE DYNATTB RESEARCH PROGRAM". en. In: Athens, Greece, pp. 4900–4910. DOI: 10.47964/1120.9397.18405. URL: https://www.easdprocedia.org/conferences/easd-conferences/eurodyn-2020/9397 (visited on Oct. 14, 2021).

Bell, Kolbein (June 2015). *Limtreboka*. en.

— (2017). *Dimensjonering av trekonstruksjoner*. ISBN: 978-82-450-2212-4.

Carmen, Sandhaas et al. (Jan. 2020). "Numerical modelling of timber and timber joints: computational aspects". English. In: *Wood Science and Technology* 54.1. Num Pages: 31-61 Place: Heidelberg, Netherlands Publisher: Springer Nature B.V., pp. 31–61. ISSN: 00437719. DOI: http://dx.doi.org/10.1007/s00226-019-01142-8. URL: https://www.proquest.com/docview/2343798501/abstract/DBC8FE589FAF4554PQ/1 (visited on Apr. 27, 2022).

CEN, European Committee for Standardization (Nov. 2004a). *Eurocode 5: Design of timber structures Part 1-1: General Common rules and rules for buildings*. European Standard. Brussels.

— (Aug. 2004b). *Eurocode 5: Design of timber structures Part 2: Bridges*. European Standard. Brussels.

— (June 2013). *NS-EN 14080:2013+NA:2016 Trekonstruksjoner Limtre og limt laminert heltre Krav*. Europeisk Standard. Standard Norge.

Dorn, Michael (2012). "Investigations on the Serviceability Limit State of Dowel-Type Timber Connections". en. In: p. 180.

Foster, Robert M., Thomas P. S. Reynolds, and Michael H. Ramage (Dec. 2016). "Proposal for Defining a Tall Timber Building". EN. In: *Journal of Structural Engineering* 142.12. Publisher: American Society of Civil Engineers, p. 02516001. ISSN: 1943-541X. DOI: 10.1061/(ASCE)ST.1943-541X.0001615. URL: https://ascelibrary.org/doi/full/10.1061/%28ASCE%29ST.1943-541X.0001615 (visited on Apr. 27, 2022).

Frette, Elisabeth, Amund Heggheim, and Thomas Munkeby (June 2021). *An experimental investigation of joints with dowels and slotted-in steel plates in glulam under service load*. Mastergrad. Norwegian University of Science and Technology Faculty of Engineering: Department of Structural Engineering.

Geiser, M., M. Bergmann, and M. Follesa (Jan. 2021). "Influence of steel properties on the ductility of doweled timber connections". en. In: *Construction and Building Materials* 266, p. 121152. ISSN: 0950-0618. DOI: 10.1016/j.conbuildmat.2020.121152. URL: https://www.sciencedirect.com/science/article/pii/S0950061820331561 (visited on Nov. 19, 2021).

Green, Michael and Jim Taggart (2020). *Tall Wood Buildings: Design, Construction and Performance. Second and Expanded Edition*. Basel/Berlin/Boston, GERMANY: Walter de

Gruyter GmbH. ISBN: 978-3-0356-1886-0. URL: http://ebookcentral.proquest.com/lib/ntnu/detail.action?docID=6132476 (visited on Apr. 27, 2022).

Malo, K. A., J. Siem, and P. Ellingsbø (Nov. 2011). "Quantifying ductility in timber structures". en. In: *Engineering Structures*. Modelling the Performance of Timber Structures 33.11, pp. 2998–3006. ISSN: 0141-0296. DOI: 10.1016/j.engstruct.2011.03.002. URL: https://www.sciencedirect.com/science/article/pii/S0141029611001210 (visited on May 5, 2022).

Malo, Kjell Arne (Aug. 2021a). "Anisotropy in Wooden Materials". en. In: p. 26.

— (May 2021b). *Quantification of energy dissipation in timber joints for low-level cyclic loading or small vibrations.* Tech. rep. NTNU Norwegian University of Science and Technology: Department of Structural Engineering.

Pletz, Martin (2022). *Efficient FE Modelling Course – understanding mechanics.* en-US. URL: https://www.martinpletz.com/fe-scripting/ (visited on May 30, 2022).

Pousette, Anna (2001). "Design values for timber bridges". en. In: p. 23.

Reynolds, Thomas, Richard Harris, and Wen-Shao Chang (Apr. 2014). "Stiffness of dowel-type timber connections under pre-yield oscillating loads". en. In: *Engineering Structures* 65, pp. 21–29. ISSN: 0141-0296. DOI: 10.1016/j.engstruct.2014.01.024. URL: https://www.sciencedirect.com/science/article/pii/S0141029614000340 (visited on Oct. 14, 2021).

Sandhaas, Carmen, Jørgen Munch-Andersen, and Philipp Dietsch (2020). *Design of Connections in Timber Structures.* A state-of-the-art report by COST Action FP1402.

Stamatopoulos, Haris (2021a). *Connections - Introduction, fasteners and examples.* en. NTNU Norwegian University of Science and Technology.

— (2021b). *Connections with laterally loaded fasteners.* en. NTNU Norwegian University of Science and Technology.

— (2021c). *Connections with steel plates and dowel-type fasteners.*

— (2021d). *Engineering wood products.* NTNU Norwegian University of Science and Technology.

UNEP, United Nations Environment Programme (2020). *2020 Global Status Report for Buildings and Construction: Towards a Zero-emission, Efficient and Resilient Buildings and Construction Sector.* Tech. rep. Nairobi.

Zonta, Daniele et al. (Mar. 2011). "Direct Displacement-Based Design of Glulam Timber Frame Buildings". en. In: *Journal of Earthquake Engineering* 15.3, pp. 491–510. ISSN: 1363-2469, 1559-808X. DOI: 10.1080/13632469.2010.495184. URL: http://www.tandfonline.com/doi/full/10.1080/13632469.2010.495184 (visited on June 7, 2022).

Appendices

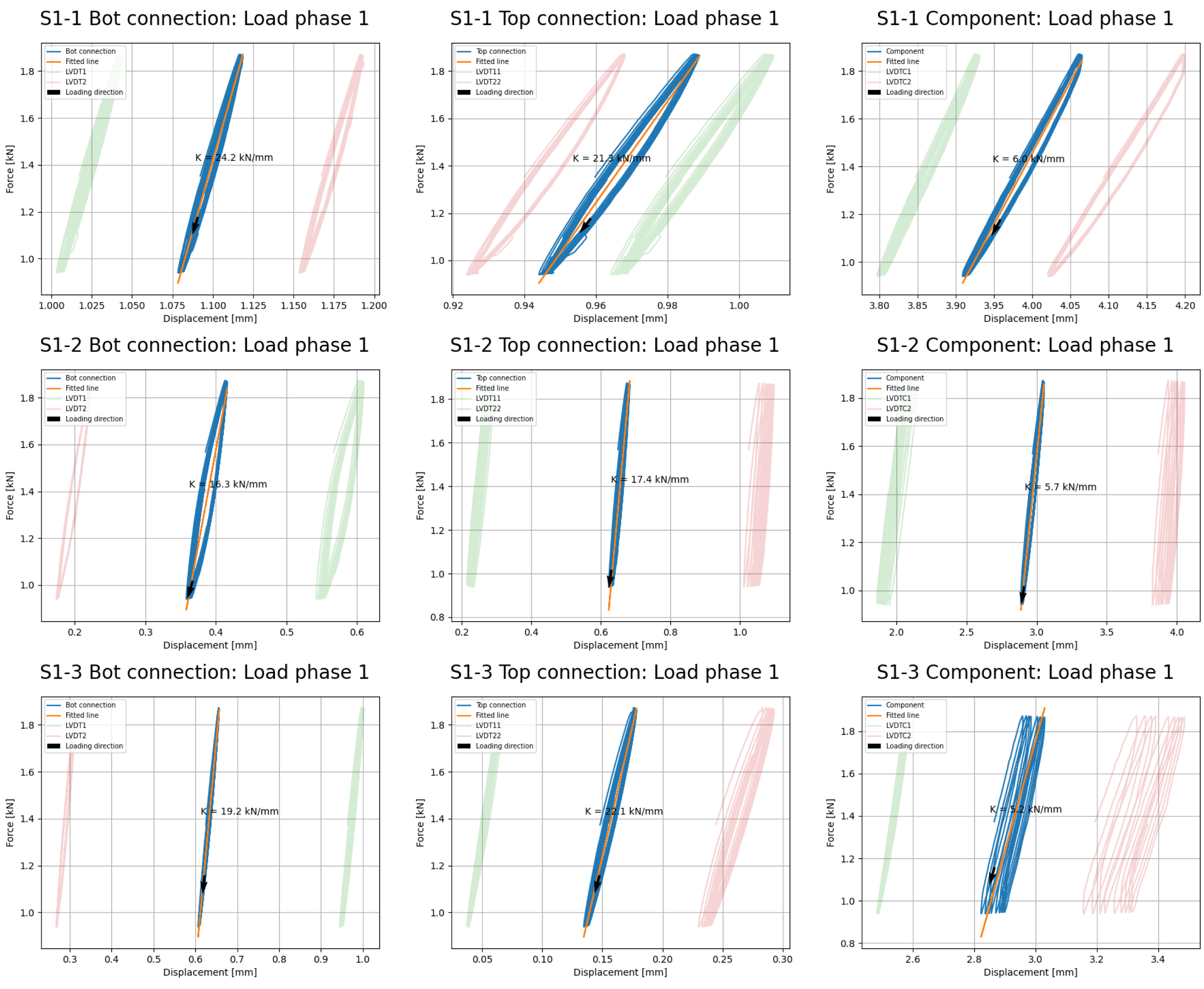
The following appendices are included:

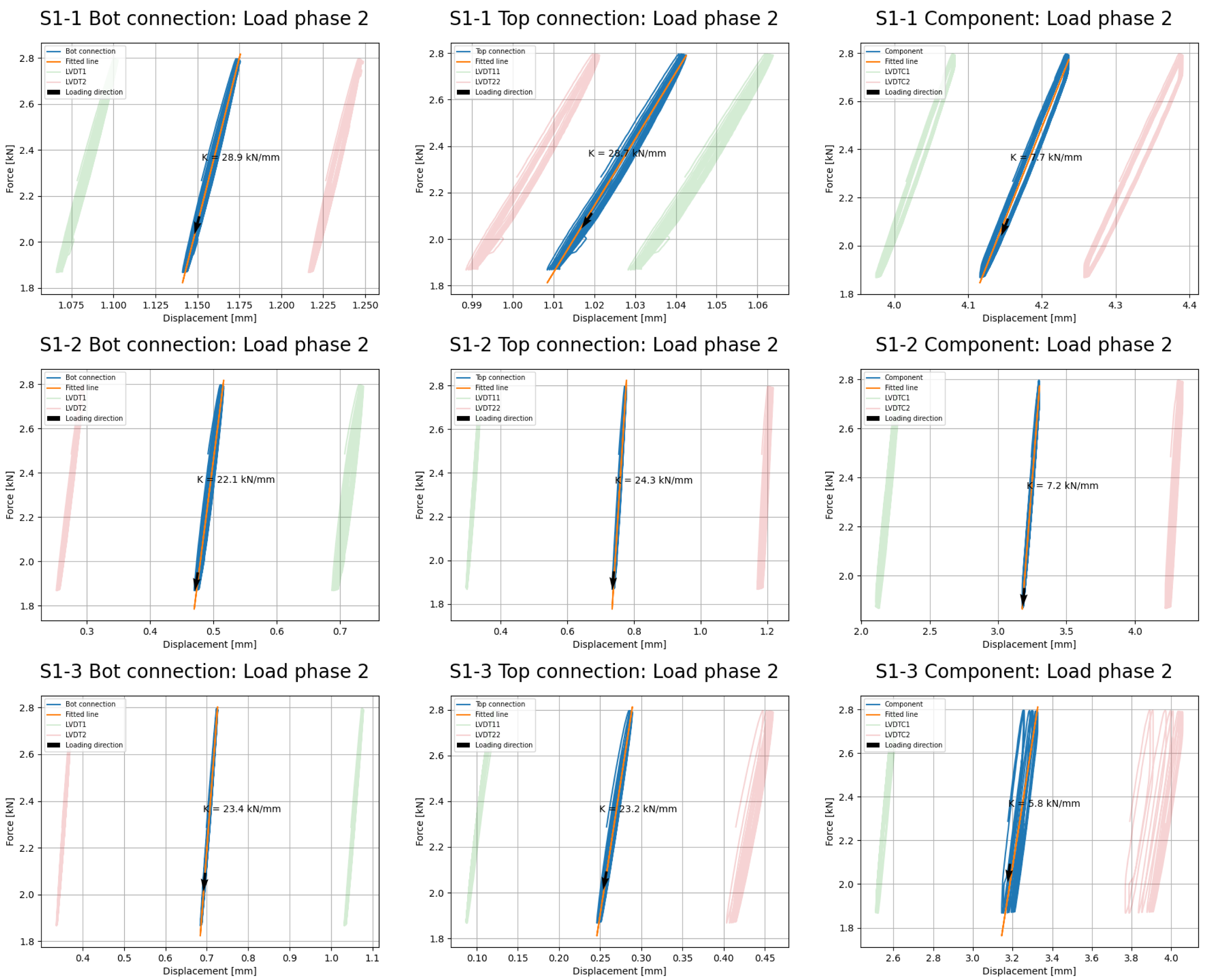
A Cyclic test results

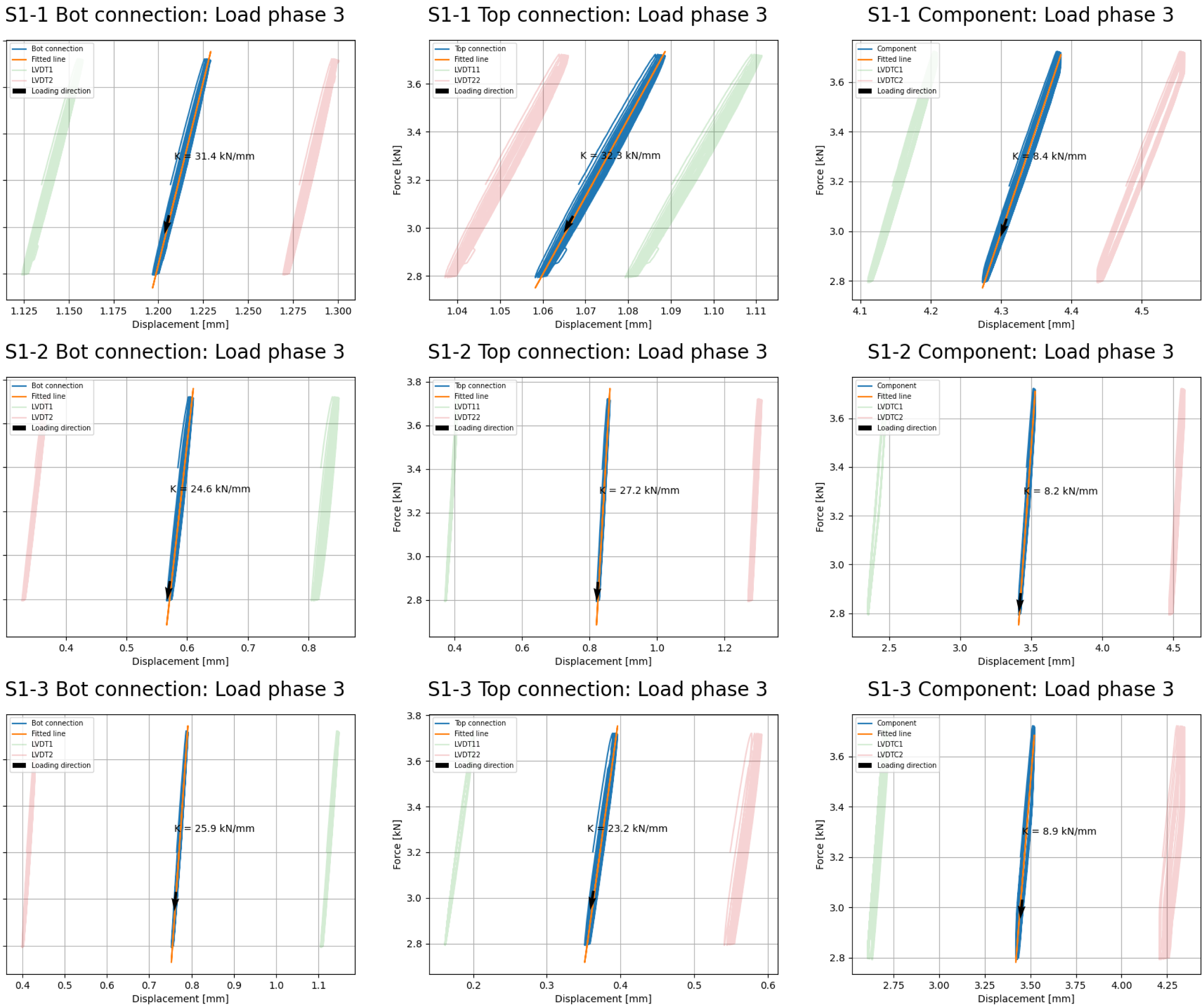
This appendix includes all hysteresis loops obtained from the cyclic testing. Each of the four load phases are presented separately. An overview is presented in table 1.

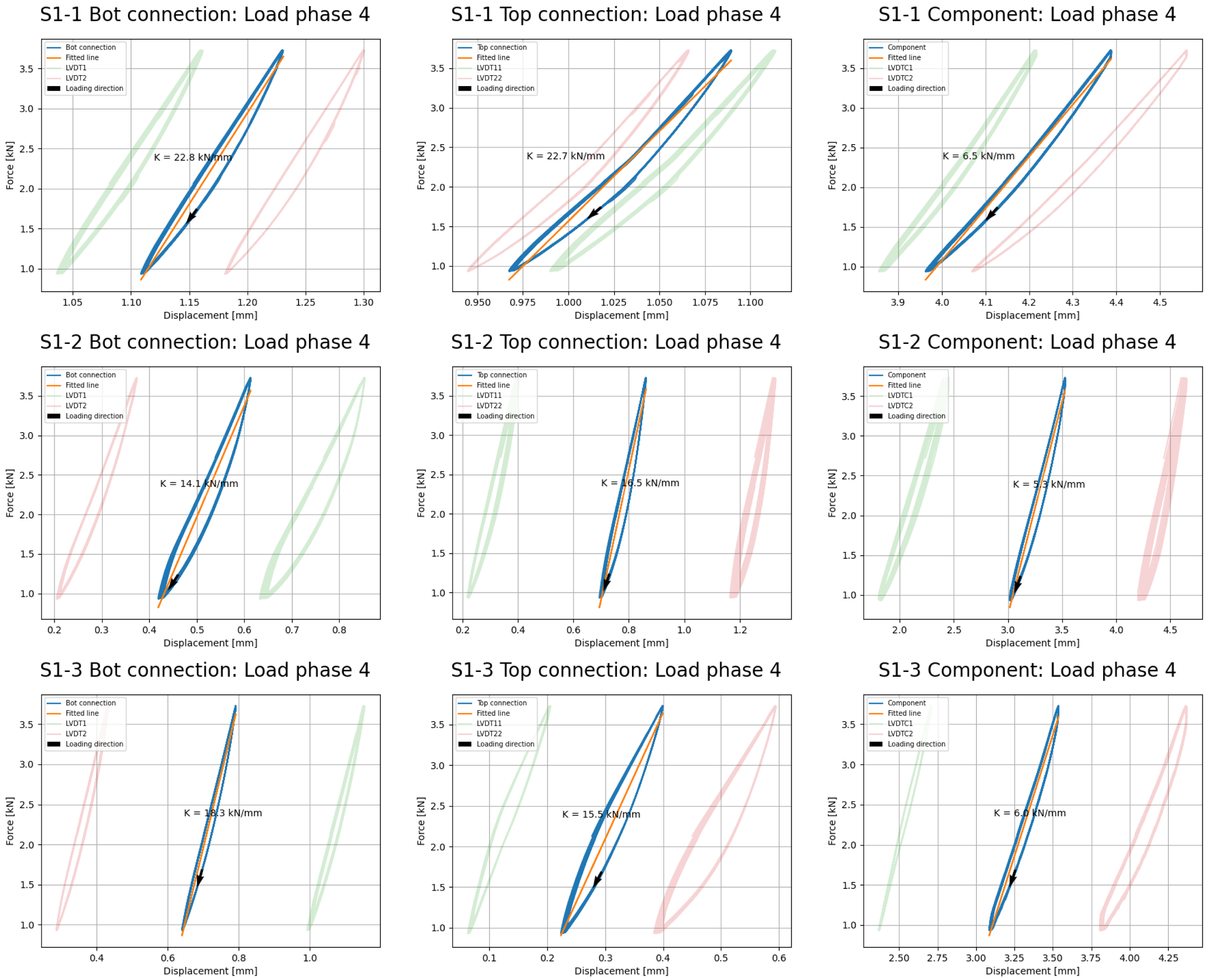
Table 1: Included documents in appendix A

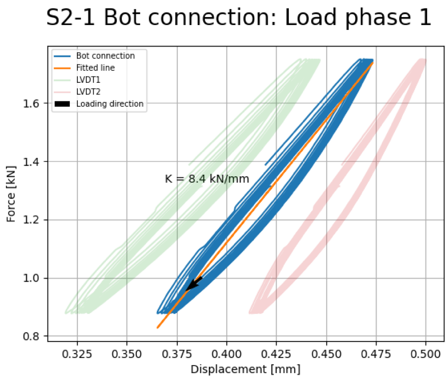
Name	Description
S1-Y, Y = 1,2,3 Load phase i, i = 1,2,3,4	Specimen type 1 was tested in the 400 kN actuator. The results are presented in term of bottom and top connection and the component measurement.
S2-Y, Y = 1,2,3,4 Load phase i, i = 1,2,3,4	Specimen type 2 was tested in the 400 kN actuator. The results are presented in term of bottom and top connection and the component measurement.
S3-Y, Y = 1,2,3 Load phase i, i = 1,2,3,4	Specimen type 3 was tested in the 100 kN actuator. The results are presented in term of bottom connection. No component measurements were done on this specimen.

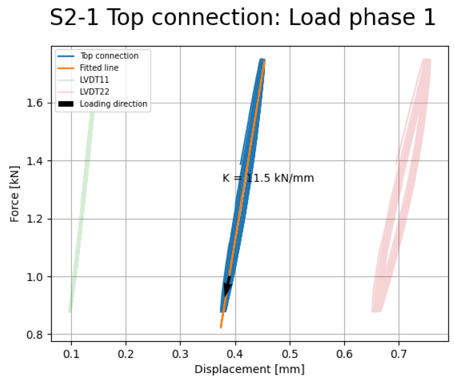

S1-1 Bot connection: Load phase 1
Bot connection
Fitted line
LVDT1
LVDT2
Loading direction
K = 24.2 kN/mm
Force [kN]
Displacement [mm]
S1-1 Top connection: Load phase 1
Top connection
Fitted line
LVDT11
LVDT22
Loading direction
K = 21.3 kN/mm
Force [kN]
Displacement [mm]
S1-1 Component: Load phase 1
Component
Fitted line
LVDTC1
LVDTC2
Loading direction
K = 6.0 kN/mm
Force [kN]
Displacement [mm]
S1-2 Bot connection: Load phase 1
K = 16.3 kN/mm
S1-2 Top connection: Load phase 1
K = 17.4 kN/mm
S1-2 Component: Load phase 1
K = 5.7 kN/mm
S1-3 Bot connection: Load phase 1
K = 19.2 kN/mm
S1-3 Top connection: Load phase 1
K = 22.1 kN/mm
S1-3 Component: Load phase 1
K = 5.2 kN/mm

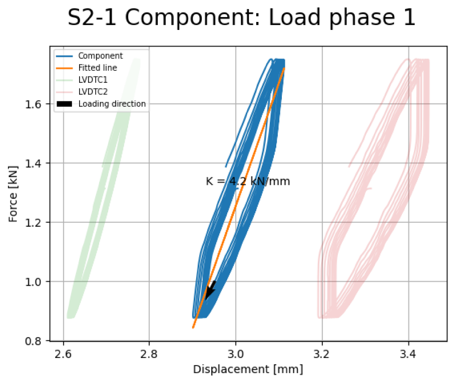

S1-1 Bot connection: Load phase 2
K = 28.9 kN/mm
S1-1 Top connection: Load phase 2
K = 28.7 kN/mm
S1-1 Component: Load phase 2
K = 7.7 kN/mm
S1-2 Bot connection: Load phase 2
K = 22.1 kN/mm
S1-2 Top connection: Load phase 2
K = 24.3 kN/mm
S1-2 Component: Load phase 2
K = 7.2 kN/mm
S1-3 Bot connection: Load phase 2
K = 23.4 kN/mm
S1-3 Top connection: Load phase 2
K = 23.2 kN/mm
S1-3 Component: Load phase 2
K = 5.8 kN/mm
Force [kN]
Displacement [mm]
Bot connection
Top connection
Component
Fitted line
LVDT1
LVDT2
LVDT11
LVDT22
LVDTC1
LVDTC2
Loading direction

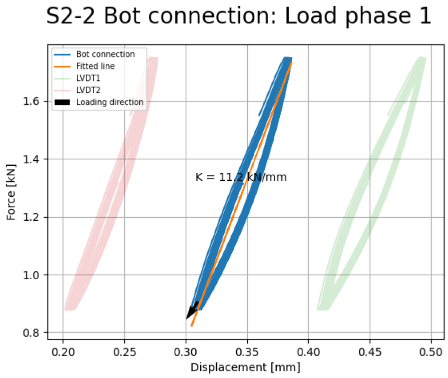

S1-1 Bot connection: Load phase 3
Bot connection
Fitted line
LVDT1
LVDT2
Loading direction
K = 31.4 kN/mm
Force [kN]
Displacement [mm]
S1-1 Top connection: Load phase 3
Top connection
Fitted line
LVDT11
LVDT22
Loading direction
K = 32.3 kN/mm
Force [kN]
Displacement [mm]
S1-1 Component: Load phase 3
Component
Fitted line
LVDTC1
LVDTC2
Loading direction
K = 8.4 kN/mm
Force [kN]
Displacement [mm]
S1-2 Bot connection: Load phase 3
Bot connection
Fitted line
LVDT1
LVDT2
Loading direction
K = 24.6 kN/mm
Force [kN]
Displacement [mm]
S1-2 Top connection: Load phase 3
Top connection
Fitted line
LVDT11
LVDT22
Loading direction
K = 27.2 kN/mm
Force [kN]
Displacement [mm]
S1-2 Component: Load phase 3
Component
Fitted line
LVDTC1
LVDTC2
Loading direction
K = 8.2 kN/mm
Force [kN]
Displacement [mm]
S1-3 Bot connection: Load phase 3
Bot connection
Fitted line
LVDT1
LVDT2
Loading direction
K = 25.9 kN/mm
Force [kN]
Displacement [mm]
S1-3 Top connection: Load phase 3
Top connection
Fitted line
LVDT11
LVDT22
Loading direction
K = 23.2 kN/mm
Force [kN]
Displacement [mm]
S1-3 Component: Load phase 3
Component
Fitted line
LVDTC1
LVDTC2
Loading direction
K = 8.9 kN/mm
Force [kN]
Displacement [mm]

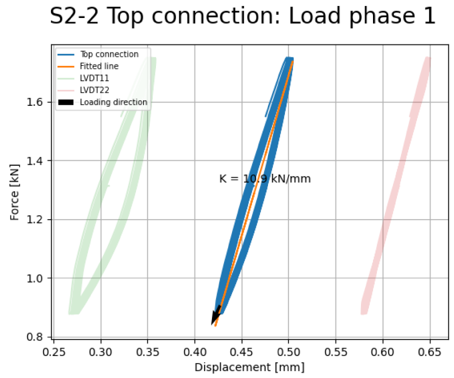

S1-1 Bot connection: Load phase 4
K = 22.8 kN/mm
S1-1 Top connection: Load phase 4
K = 22.7 kN/mm
S1-1 Component: Load phase 4
K = 6.5 kN/mm
S1-2 Bot connection: Load phase 4
K = 14.1 kN/mm
S1-2 Top connection: Load phase 4
K = 16.5 kN/mm
S1-2 Component: Load phase 4
K = 5.3 kN/mm
S1-3 Bot connection: Load phase 4
K = 18.3 kN/mm
S1-3 Top connection: Load phase 4
K = 15.5 kN/mm
S1-3 Component: Load phase 4
K = 6.0 kN/mm
Force [kN]
Displacement [mm]
Bot connection
Top connection
Component
Fitted line
LVDT1
LVDT2
LVDT11
LVDT22
LVDTC1
LVDTC2
Loading direction

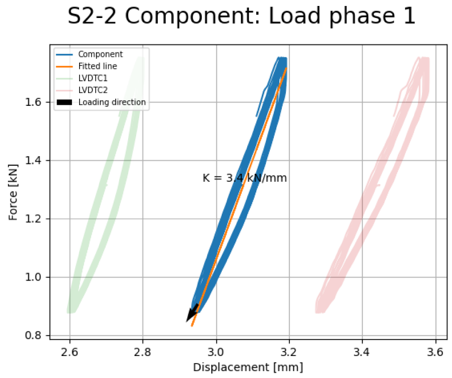

S2-1 Bot connection: Load phase 1
Bot connection
Fitted line
LVDT1
LVDT2
Loading direction
K = 8.4 kN/mm
Force [kN]
Displacement [mm]

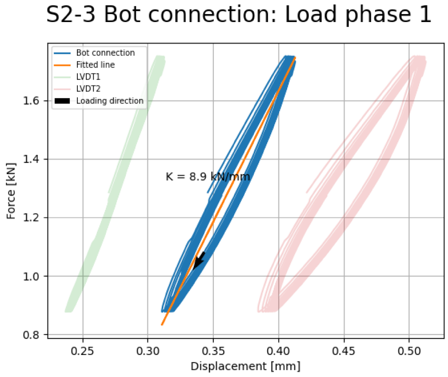

S2-1 Top connection: Load phase 1
Top connection
Fitted line
LVDT11
LVDT22
Loading direction
K = 11.5 kN/mm
Force [kN]
Displacement [mm]

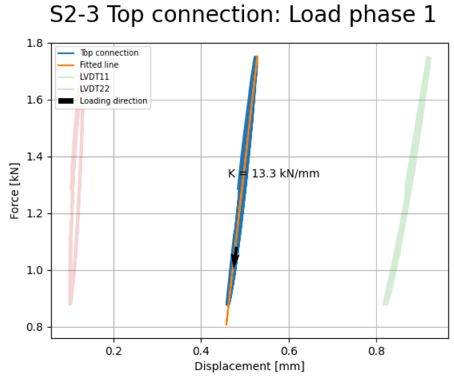

S2-1 Component: Load phase 1
Component
Fitted line
LVDTC1
LVDTC2
Loading direction
K = 4.2 kN/mm
Force [kN]
Displacement [mm]

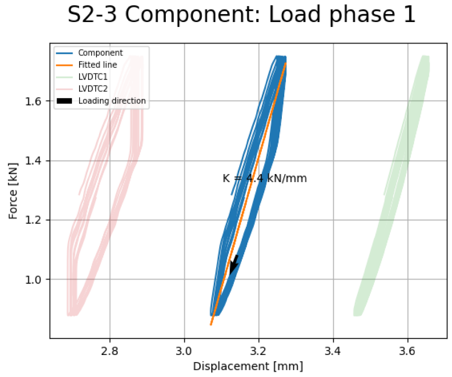

S2-2 Bot connection: Load phase 1
Bot connection
Fitted line
LVDT1
LVDT2
Loading direction
K = 11.2 kN/mm
Force [kN]
Displacement [mm]

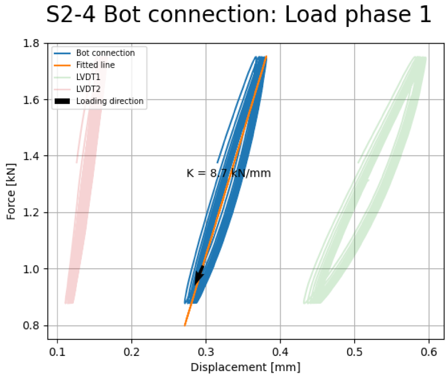

S2-2 Top connection: Load phase 1
Top connection
Fitted line
LVDT11
LVDT22
Loading direction
K = 10.9 kN/mm
Force [kN]
Displacement [mm]

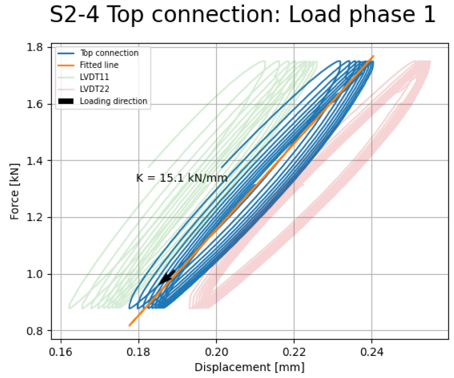

S2-2 Component: Load phase 1
Component
Fitted line
LVDTC1
LVDTC2
Loading direction
K = 3.4 kN/mm
Force [kN]
Displacement [mm]

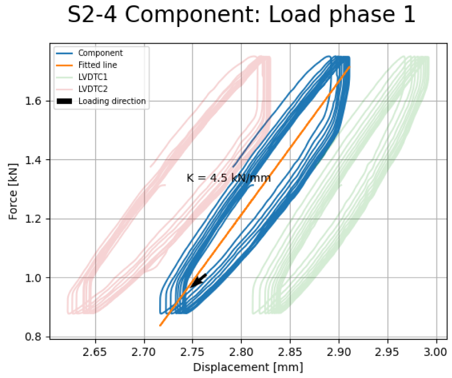

S2-3 Bot connection: Load phase 1
Bot connection
Fitted line
LVDT1
LVDT2
Loading direction
K = 8.9 kN/mm
Force [kN]
Displacement [mm]

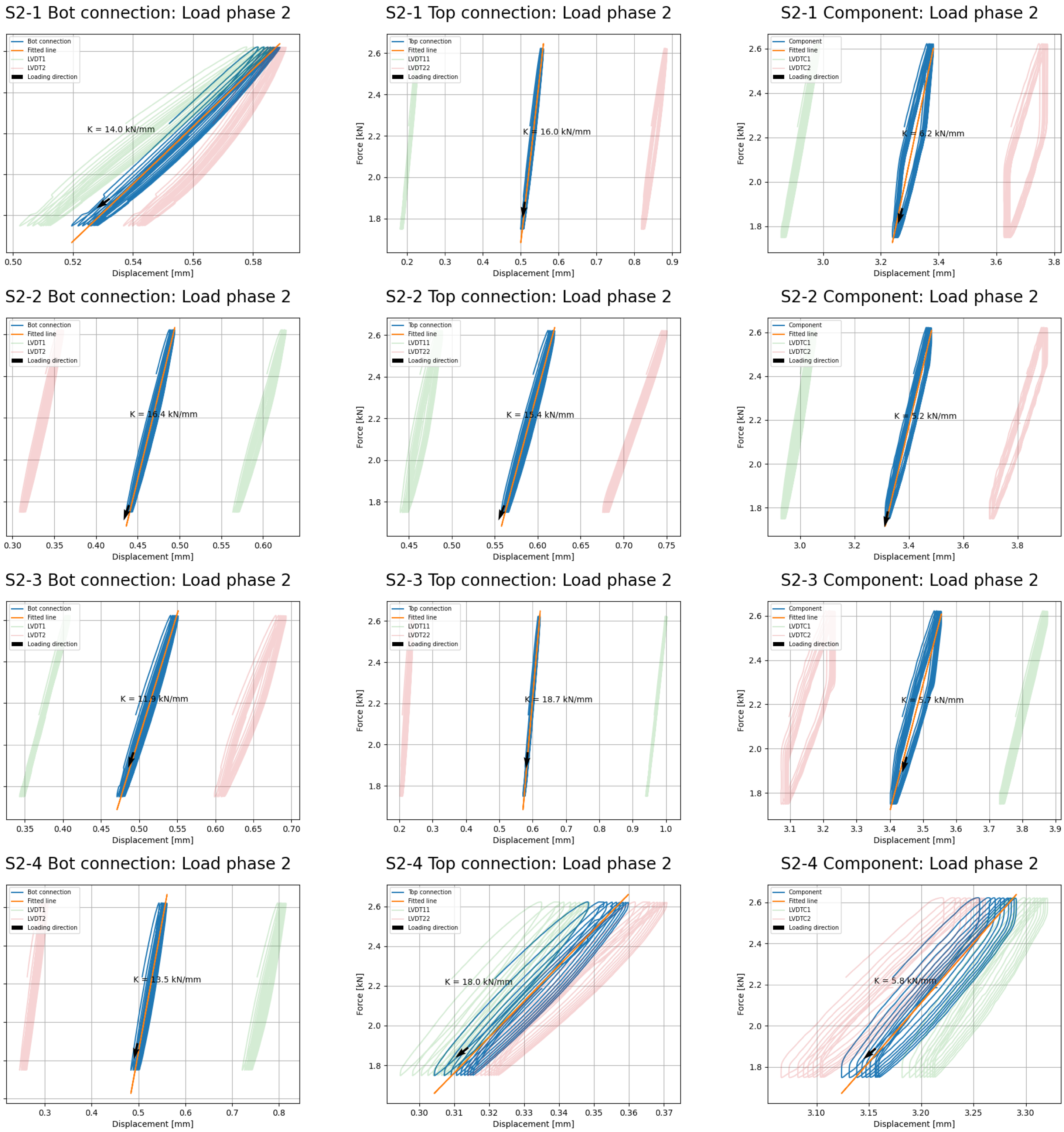

S2-3 Top connection: Load phase 1
Top connection
Fitted line
LVDT11
LVDT22
Loading direction
K = 13.3 kN/mm
Force [kN]
Displacement [mm]


S2-3 Component: Load phase 1
Component
Fitted line
LVDTC1
LVDTC2
Loading direction
K = 4.4 kN/mm
Force [kN]
Displacement [mm]

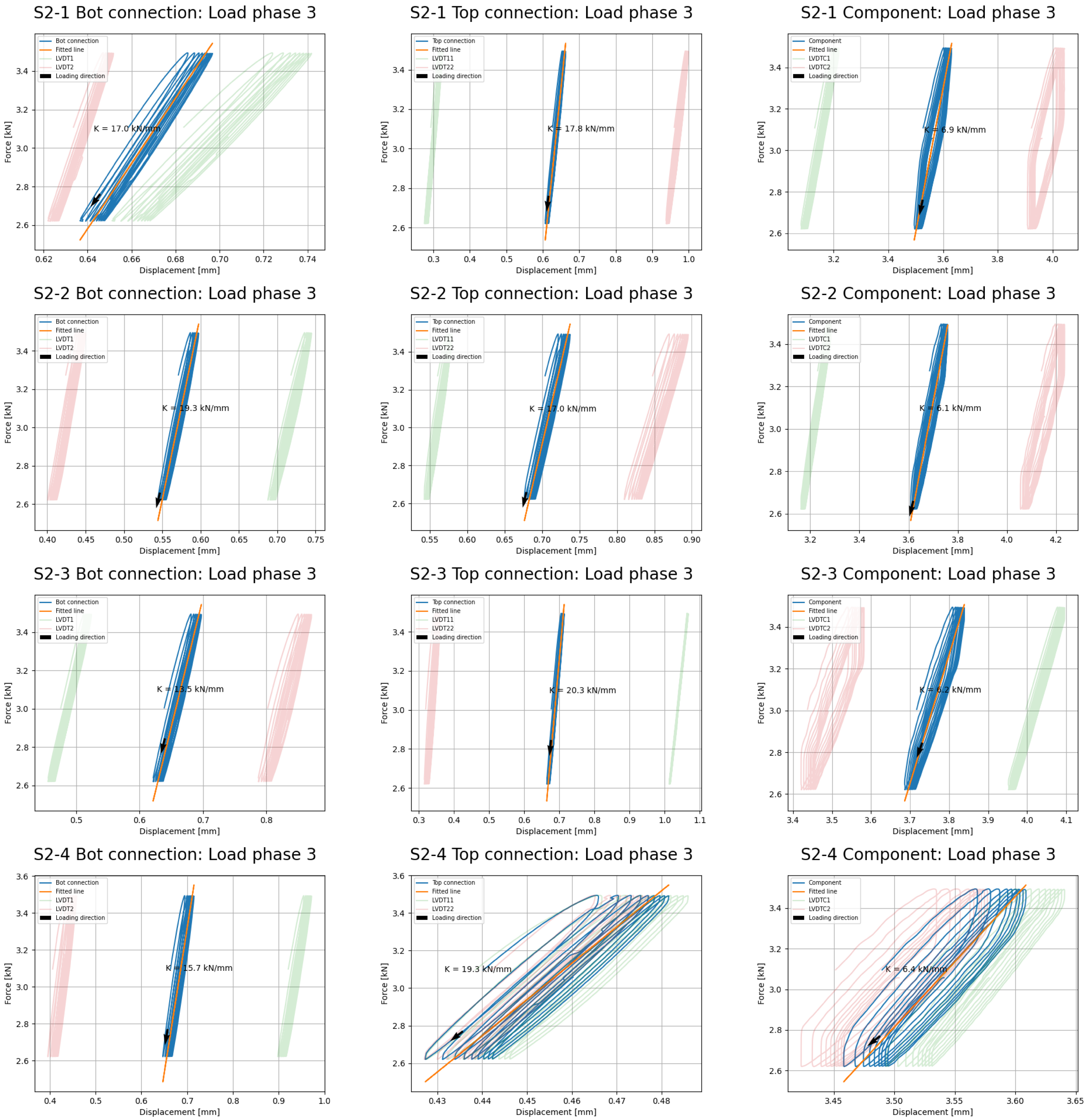

S2-4 Bot connection: Load phase 1
Bot connection
Fitted line
LVDT1
LVDT2
Loading direction
K = 8.7 kN/mm
Force [kN]
Displacement [mm]


S2-4 Top connection: Load phase 1
Top connection
Fitted line
LVDT11
LVDT22
Loading direction
K = 15.1 kN/mm
Force [kN]
Displacement [mm]

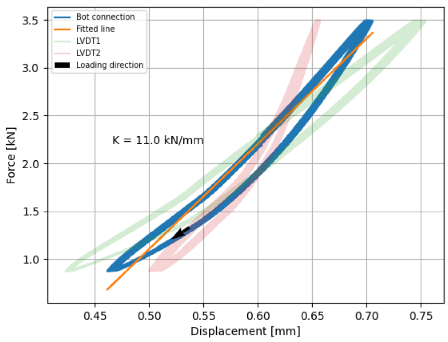

S2-4 Component: Load phase 1
Component
Fitted line
LVDTC1
LVDTC2
Loading direction
K = 4.5 kN/mm
Force [kN]
Displacement [mm]



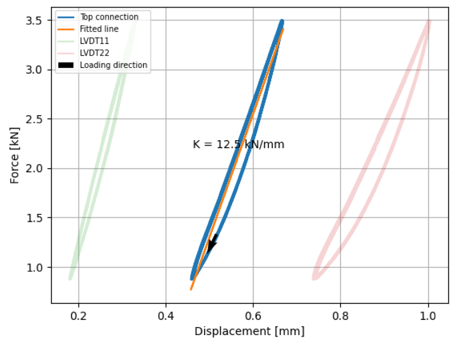
S2-1 Bot connection: Load phase 2
K = 14.0 kN/mm
S2-1 Top connection: Load phase 2
K = 16.0 kN/mm
S2-1 Component: Load phase 2
K = 6.2 kN/mm
S2-2 Bot connection: Load phase 2
K = 16.4 kN/mm
S2-2 Top connection: Load phase 2
K = 15.4 kN/mm
S2-2 Component: Load phase 2
K = 5.2 kN/mm
S2-3 Bot connection: Load phase 2
K = 11.9 kN/mm
S2-3 Top connection: Load phase 2
K = 18.7 kN/mm
S2-3 Component: Load phase 2
K = 5.7 kN/mm
S2-4 Bot connection: Load phase 2
K = 13.5 kN/mm
S2-4 Top connection: Load phase 2
K = 18.0 kN/mm
S2-4 Component: Load phase 2
K = 5.8 kN/mm
Force [kN]
Displacement [mm]
Bot connection
Top connection
Component
Fitted line
LVDT1
LVDT2
LVDT11
LVDT22
LVDTC1
LVDTC2
Loading direction


S2-1 Bot connection: Load phase 3
K = 17.0 kN/mm
S2-1 Top connection: Load phase 3
K = 17.8 kN/mm
S2-1 Component: Load phase 3
K = 6.9 kN/mm
S2-2 Bot connection: Load phase 3
K = 19.3 kN/mm
S2-2 Top connection: Load phase 3
K = 17.0 kN/mm
S2-2 Component: Load phase 3
K = 6.1 kN/mm
S2-3 Bot connection: Load phase 3
K = 13.5 kN/mm
S2-3 Top connection: Load phase 3
K = 20.3 kN/mm
S2-3 Component: Load phase 3
K = 6.2 kN/mm
S2-4 Bot connection: Load phase 3
K = 15.7 kN/mm
S2-4 Top connection: Load phase 3
K = 19.3 kN/mm
S2-4 Component: Load phase 3
K = 6.4 kN/mm
Force [kN]
Displacement [mm]
Bot connection
Top connection
Component
Fitted line
LVDT1
LVDT2
LVDT11
LVDT22
LVDTC1
LVDTC2
Loading direction

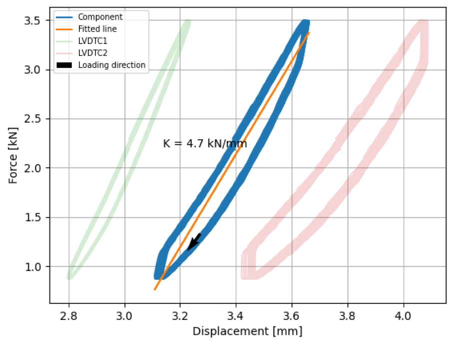
S2-1 Bot connection: Load phase 4



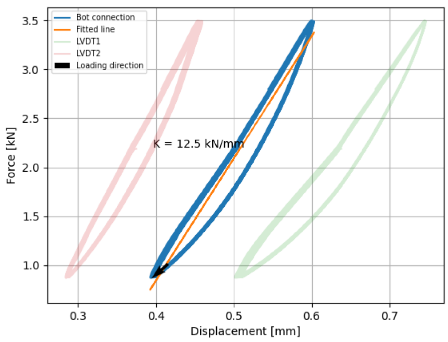
S2-1 Top connection: Load phase 4



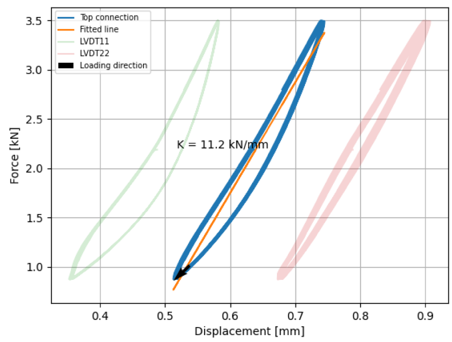
S2-1 Component: Load phase 4



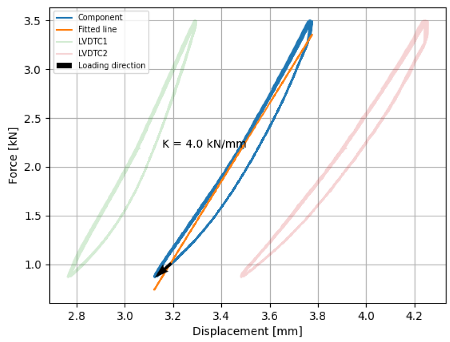
S2-2 Bot connection: Load phase 4



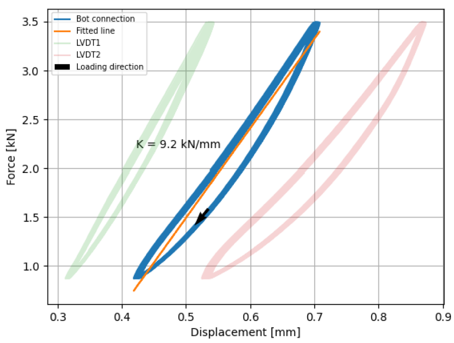
S2-2 Top connection: Load phase 4



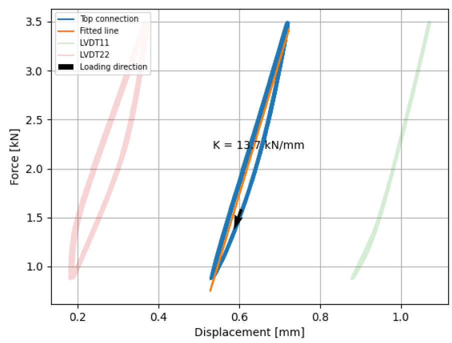
S2-2 Component: Load phase 4



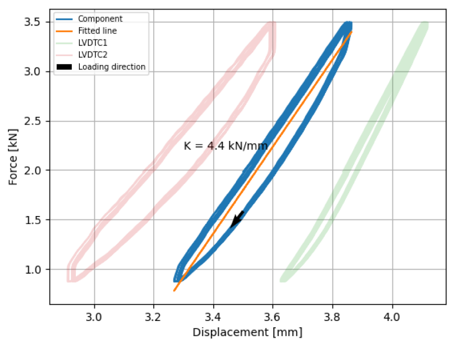
S2-3 Bot connection: Load phase 4



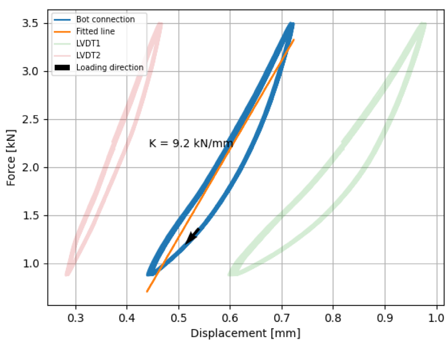
S2-3 Top connection: Load phase 4



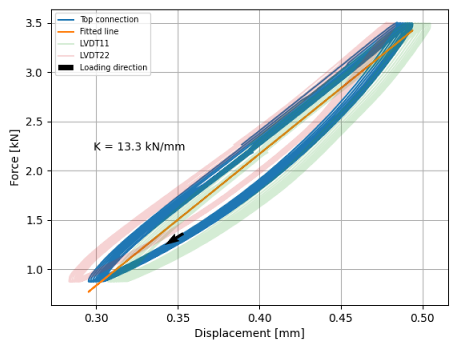
S2-3 Component: Load phase 4



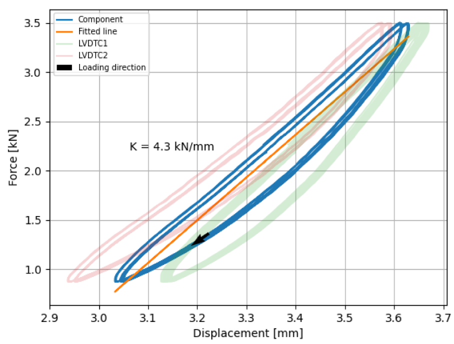
S2-4 Bot connection: Load phase 4



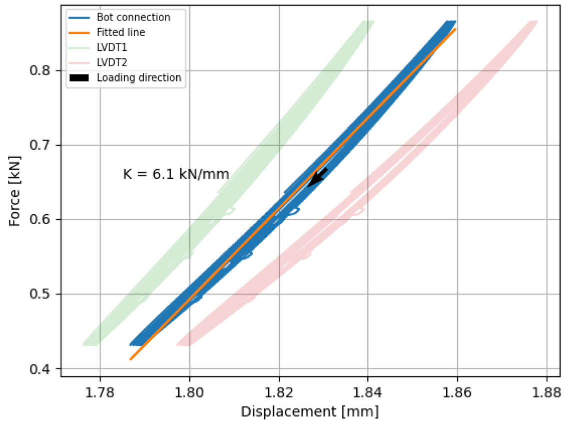
S2-4 Top connection: Load phase 4



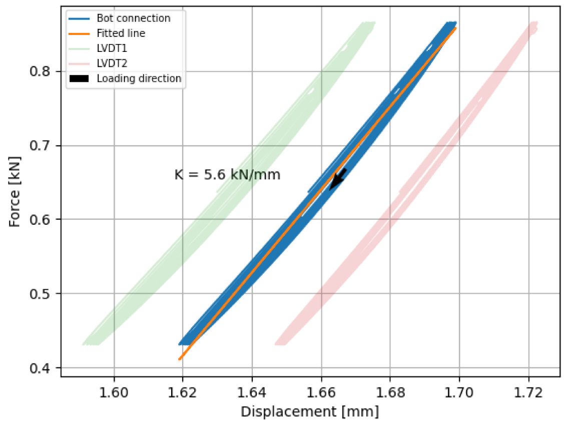
S2-4 Component: Load phase 4



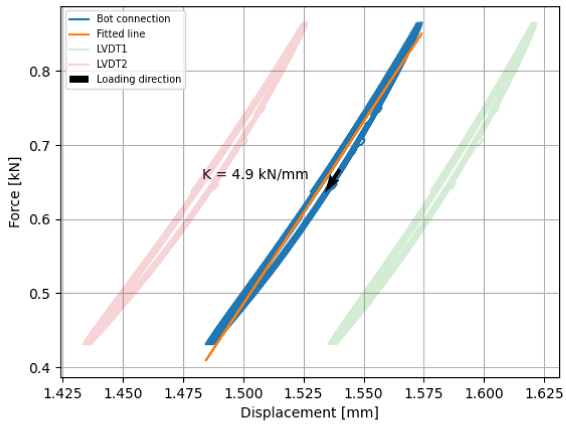
S3-1 Bot connection: Load phase 1

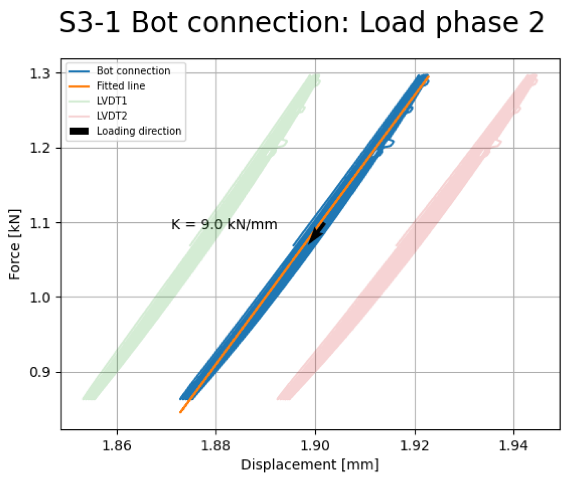


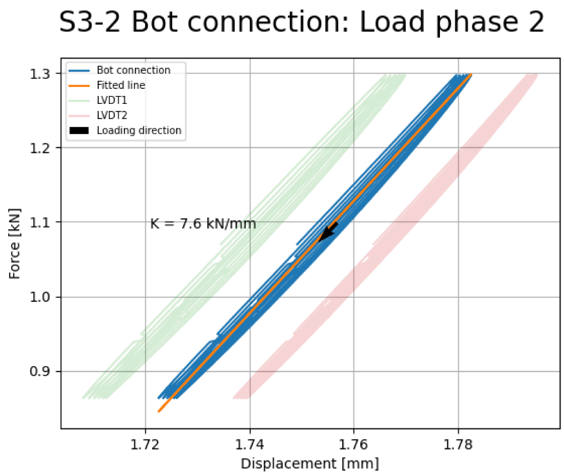
S3-2 Bot connection: Load phase 1

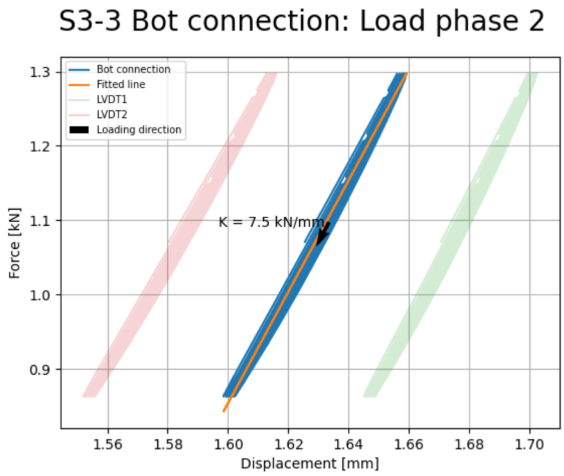


S3-3 Bot connection: Load phase 1

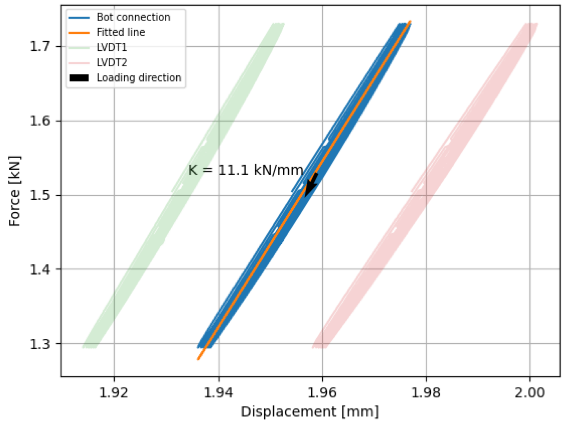



S3-1 Bot connection: Load phase 2
Bot connection
Fitted line
LVDT1
LVDT2
Loading direction
K = 9.0 kN/mm
Force [kN]
Displacement [mm]
1.3
1.2
1.1
1.0
0.9
1.86
1.88
1.90
1.92
1.94

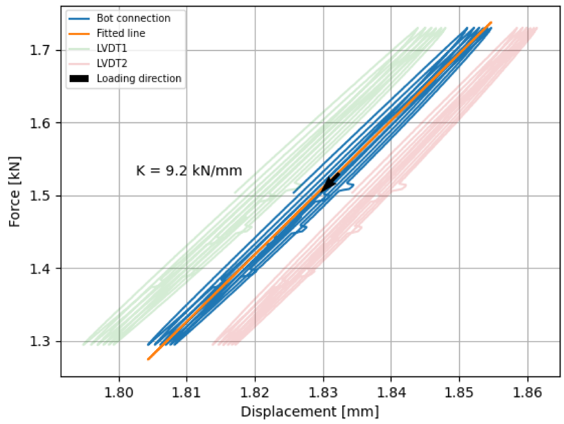

S3-2 Bot connection: Load phase 2
Bot connection
Fitted line
LVDT1
LVDT2
Loading direction
K = 7.6 kN/mm
Force [kN]
Displacement [mm]
1.3
1.2
1.1
1.0
0.9
1.72
1.74
1.76
1.78


S3-3 Bot connection: Load phase 2
Bot connection
Fitted line
LVDT1
LVDT2
Loading direction
K = 7.5 kN/mm
Force [kN]
Displacement [mm]
1.3
1.2
1.1
1.0
0.9
1.56
1.58
1.60
1.62
1.64
1.66
1.68
1.70

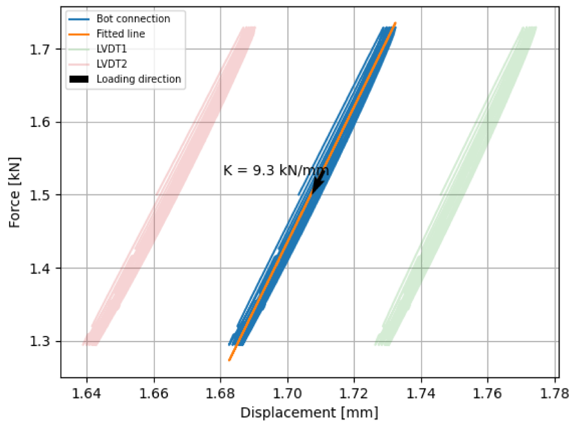
S3-1 Bot connection: Load phase 3

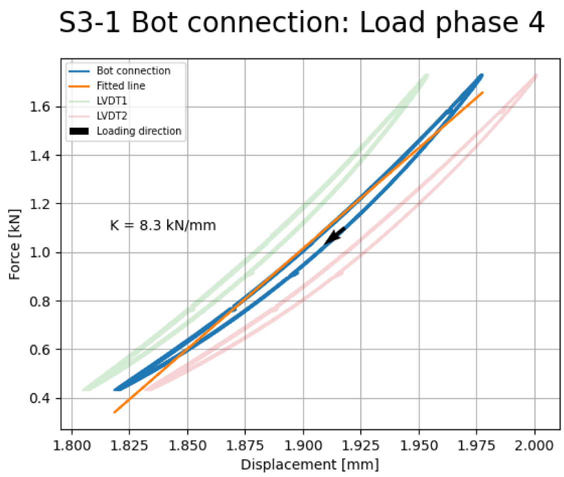


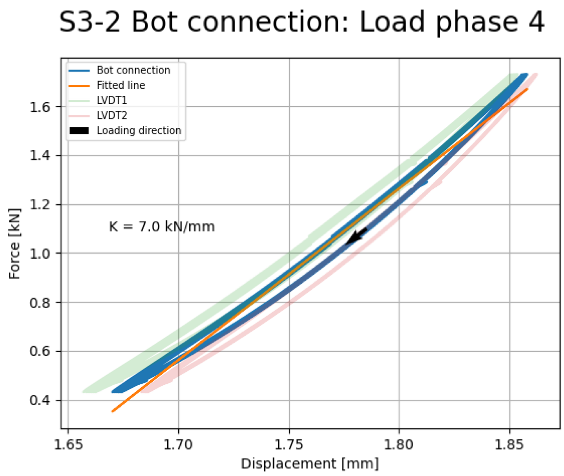
S3-2 Bot connection: Load phase 3

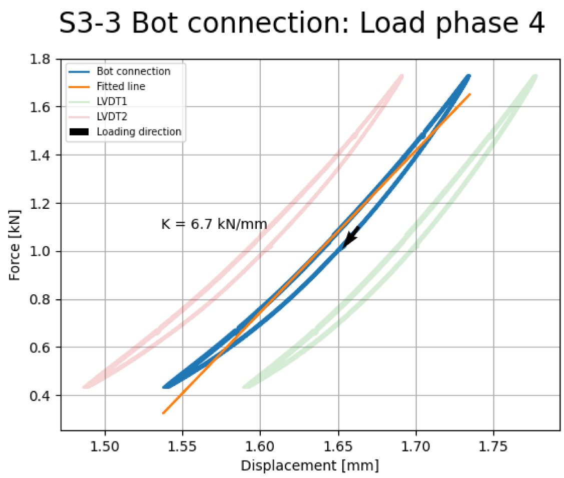


S3-3 Bot connection: Load phase 3




S3-1 Bot connection: Load phase 4
Bot connection
Fitted line
LVDT1
LVDT2
Loading direction
K = 8.3 kN/mm
Force [kN]
Displacement [mm]
1.800
1.825
1.850
1.875
1.900
1.925
1.950
1.975
2.000
0.4
0.6
0.8
1.0
1.2
1.4
1.6


S3-2 Bot connection: Load phase 4
Bot connection
Fitted line
LVDT1
LVDT2
Loading direction
K = 7.0 kN/mm
Force [kN]
Displacement [mm]
1.65
1.70
1.75
1.80
1.85
0.4
0.6
0.8
1.0
1.2
1.4
1.6


S3-3 Bot connection: Load phase 4
Bot connection
Fitted line
LVDT1
LVDT2
Loading direction
K = 6.7 kN/mm
Force [kN]
Displacement [mm]
1.50
1.55
1.60
1.65
1.70
1.75
0.4
0.6
0.8
1.0
1.2
1.4
1.6
1.8

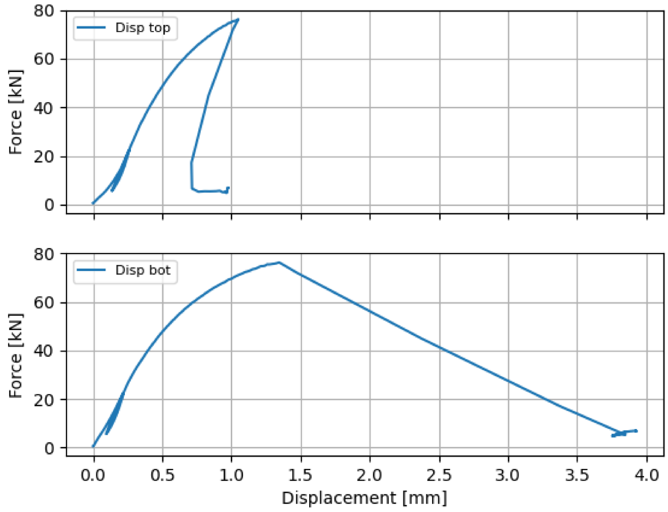
B Failure test results

This appendix includes the plots provided from the failure test data series. An overview is presented in table 2.

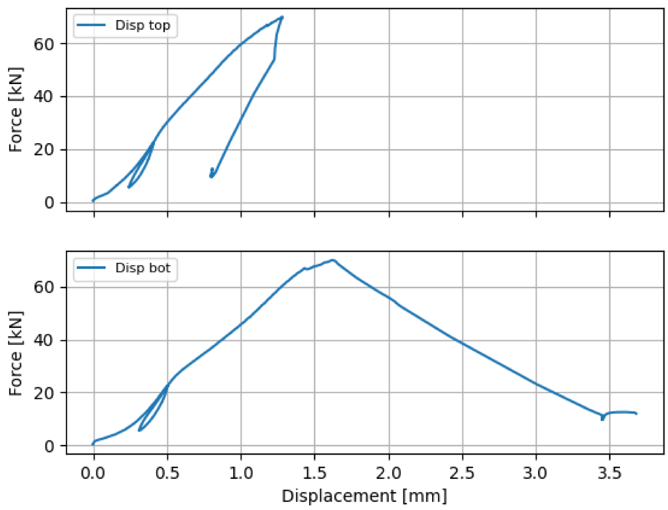
Table 2: Included documents in appendix B

Name	Description
S1-Y, Y = 1,2,3	Specimen type 1 was ran till failure in the 400 kN actuator. The results are presented in terms of bottom and top connection as force-displacement curves.
S2-Y, Y = 1,2,3,4	Specimen type 2 was ran till failure in the 400 kN actuator. The results are presented in terms of bottom and top connection as force-displacement curves.
S3-Y, Y = 1,2,3	Specimen type 3 was ran till failure in the 100 kN actuator. The results are presented in term of bottom connection as force-displacement curves.

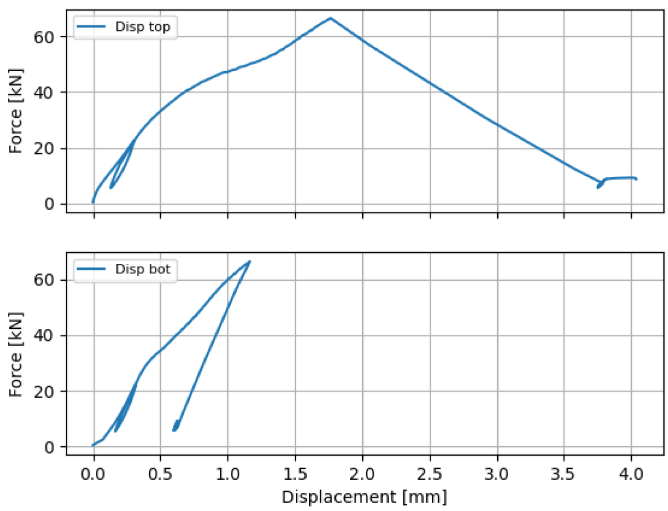
S1-1 Failure test

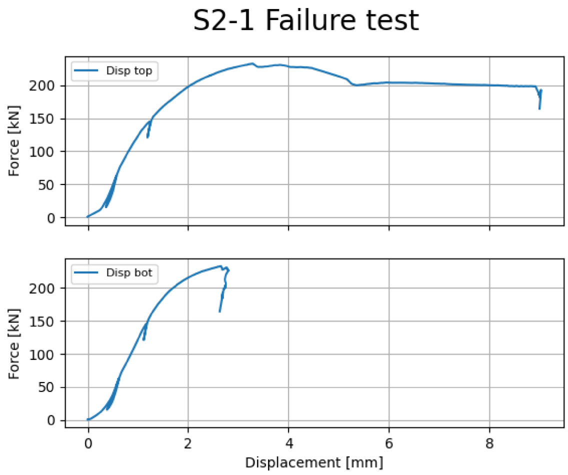


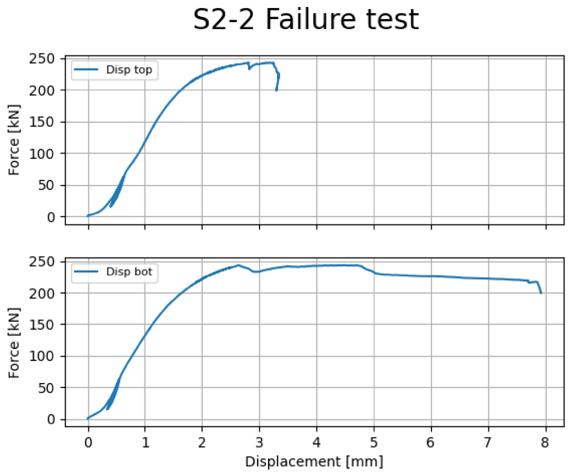
S1-2 Failure test

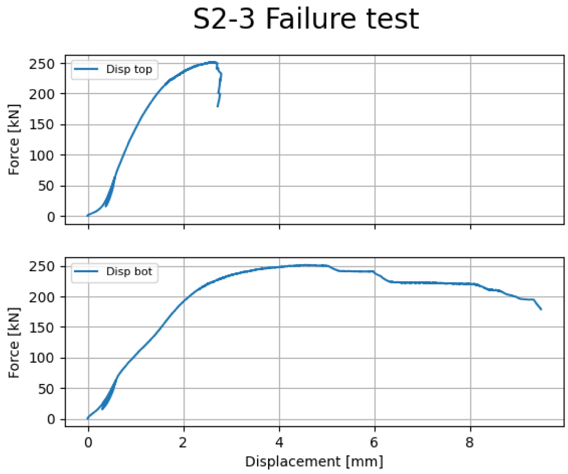


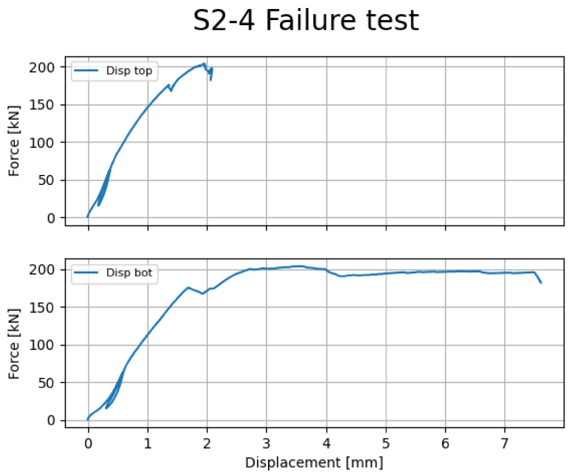
S1-3 Failure test

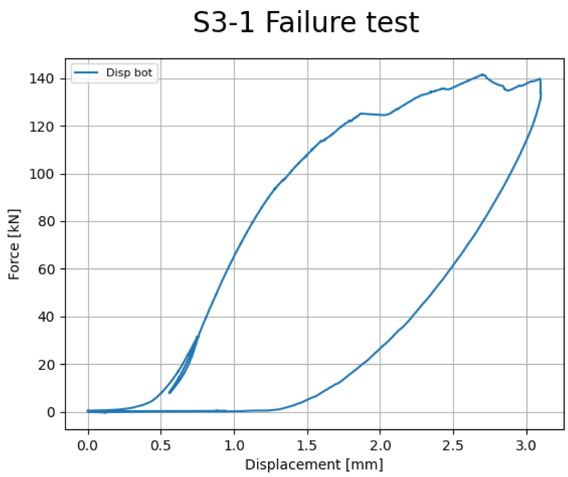


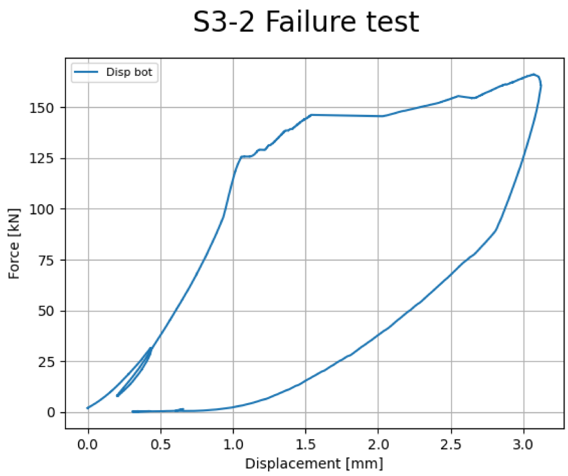

S2-1 Failure test
Disp top
Disp bot
Force [kN]
Displacement [mm]


S2-2 Failure test
Disp top
Disp bot
Force [kN]
Displacement [mm]


S2-3 Failure test
Disp top
Disp bot
Force [kN]
Displacement [mm]


S2-4 Failure test
Disp top
Disp bot
Force [kN]
Displacement [mm]

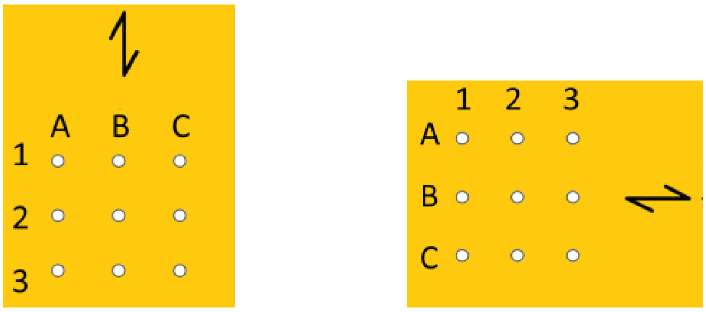

S3-1 Failure test
Disp bot
Force [kN]
Displacement [mm]


S3-2 Failure test
Disp bot
Force [kN]
Displacement [mm]


S3-3 Failure test
Disp bot
Force [kN]
Displacement [mm]

C Zero and full stiffness results (Frette et al., 2021)

This appendix includes all hysteresis loops obtained from the data series of Frette et al. (2021) for fully reversed loading. The naming of each plot follows the notation system of the configurations as described in section 3.1.5.1. A brief recap of the configuration system is provided in figure 1. An overview of included documents follows in table 3.

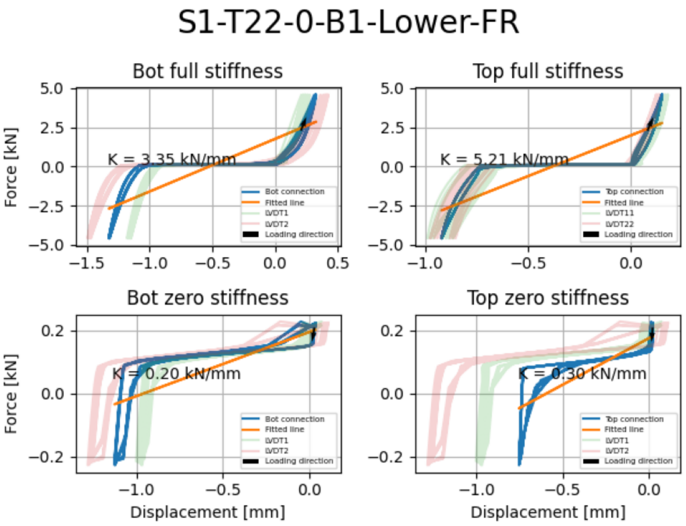


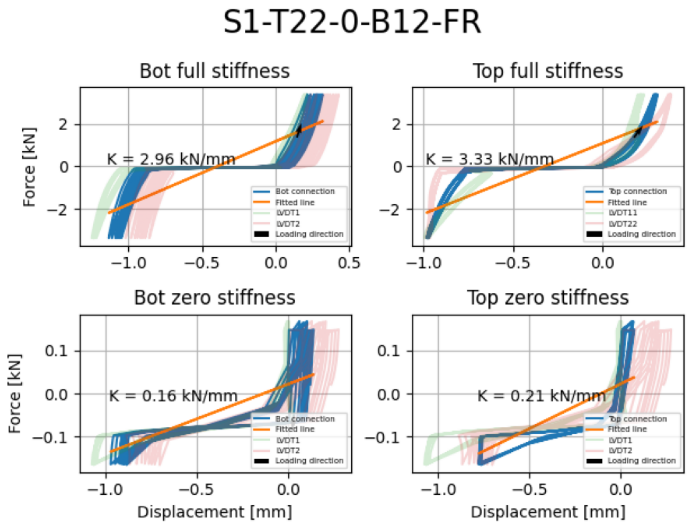
(a) S1 and S2 (b) S3

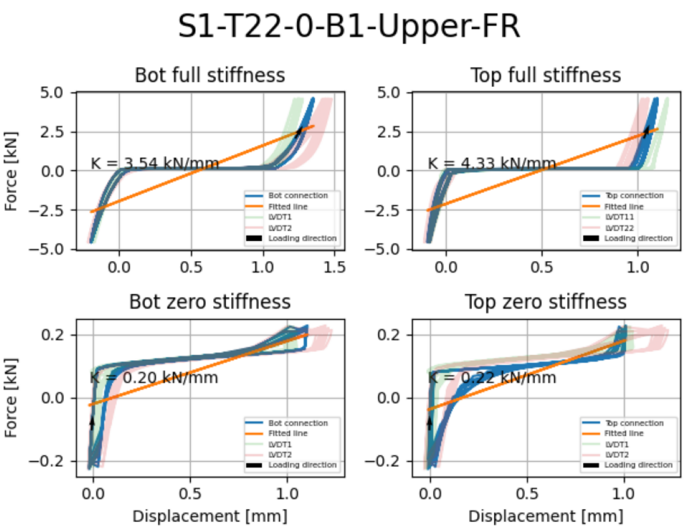
Figure 1: Configuration basis

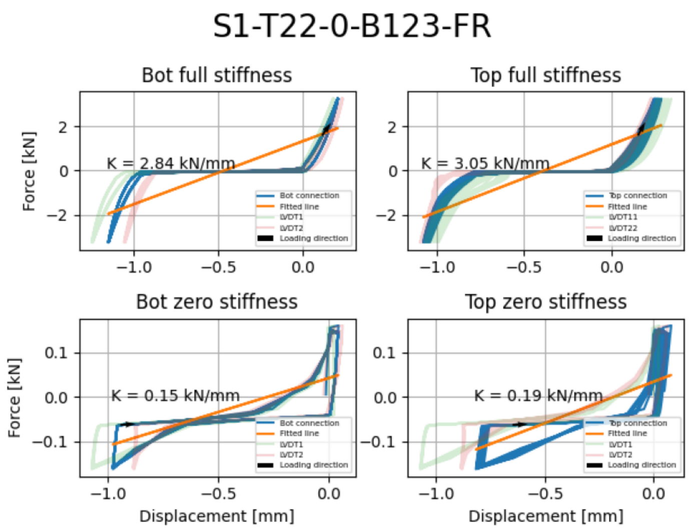
Table 3: Included documents in appendix C.

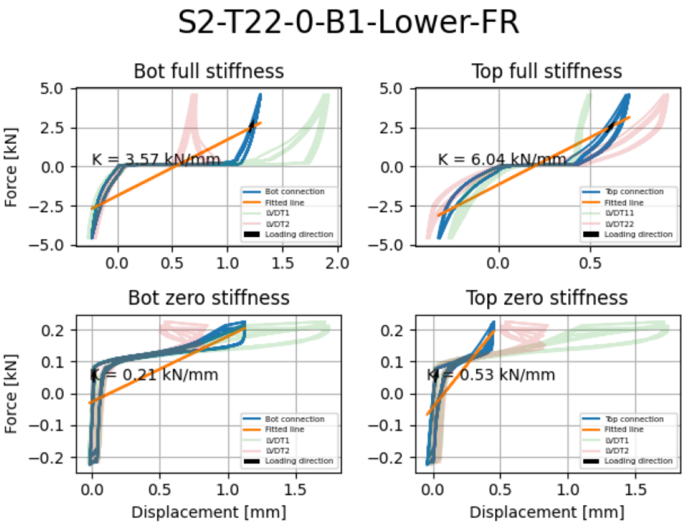
Name	**Description**
SY-T22-00-Configuration-FR, Y = 1,2,3	Results for specimen type 1 exposed to fully reversed loading based on data from Frette. The results are presented in terms of bottom/top connection full and zero stiffness.
SY-T15-00-Configuration-FR, Y = 4,5,6	Results for specimen type 2 exposed to fully reversed loading based on data from Frette. The results are presented in terms of bottom/top connection full and zero stiffness.
SY-T15-90-Configuration-FR, Y = 7,8,9	Results for specimen type 3 exposed to fully reversed loading based on data from Frette. The results are presented in terms of bottom connection full and zero stiffness.

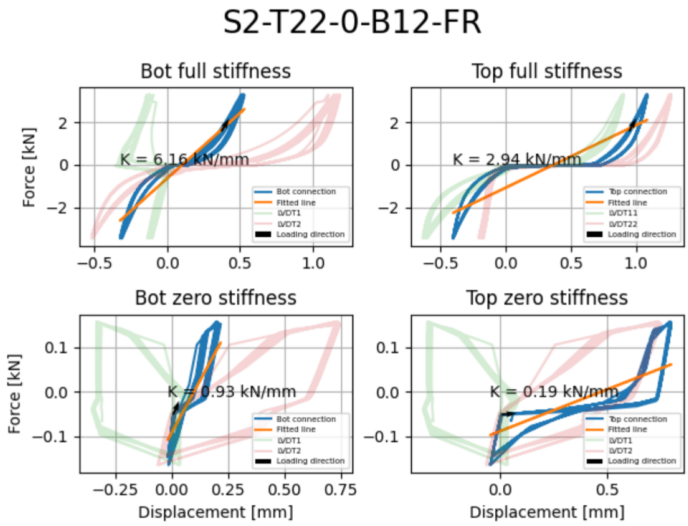

S1-T22-0-B1-Lower-FR
Bot full stiffness
Top full stiffness
K = 3.35 kN/mm
K = 5.21 kN/mm
Bot zero stiffness
Top zero stiffness
K = 0.20 kN/mm
K = 0.30 kN/mm
Force [kN]
Displacement [mm]

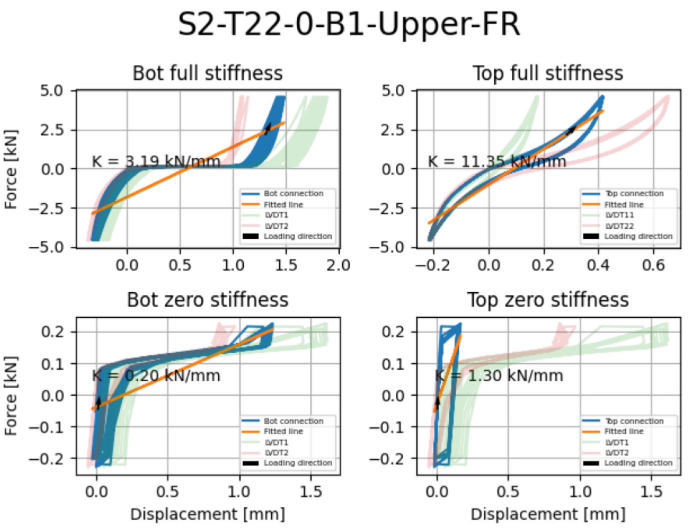

S1-T22-0-B12-FR
Bot full stiffness
Top full stiffness
K = 2.96 kN/mm
K = 3.33 kN/mm
Bot zero stiffness
Top zero stiffness
K = 0.16 kN/mm
K = 0.21 kN/mm
Force [kN]
Displacement [mm]

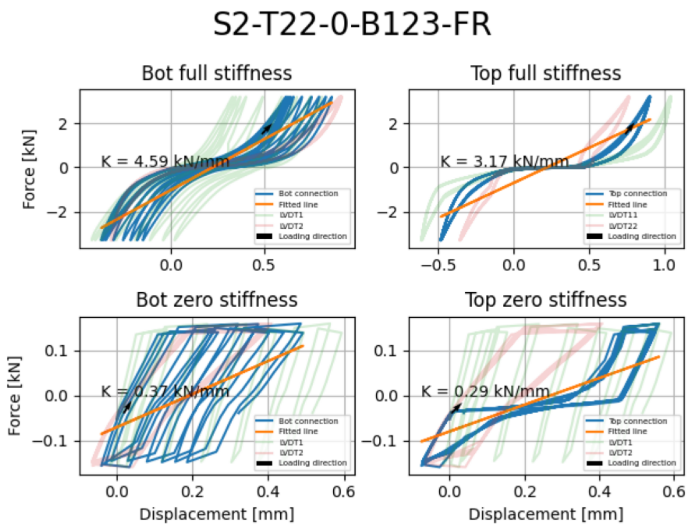

S1-T22-0-B1-Upper-FR
Bot full stiffness
Top full stiffness
K = 3.54 kN/mm
K = 4.33 kN/mm
Bot zero stiffness
Top zero stiffness
K = 0.20 kN/mm
K = 0.22 kN/mm
Force [kN]
Displacement [mm]

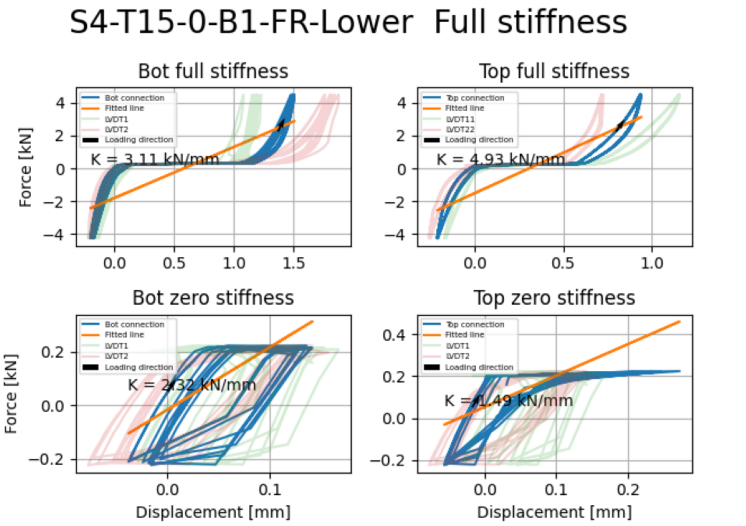

S1-T22-0-B123-FR
Bot full stiffness
Top full stiffness
K = 2.84 kN/mm
K = 3.05 kN/mm
Bot zero stiffness
Top zero stiffness
K = 0.15 kN/mm
K = 0.19 kN/mm
Force [kN]
Displacement [mm]

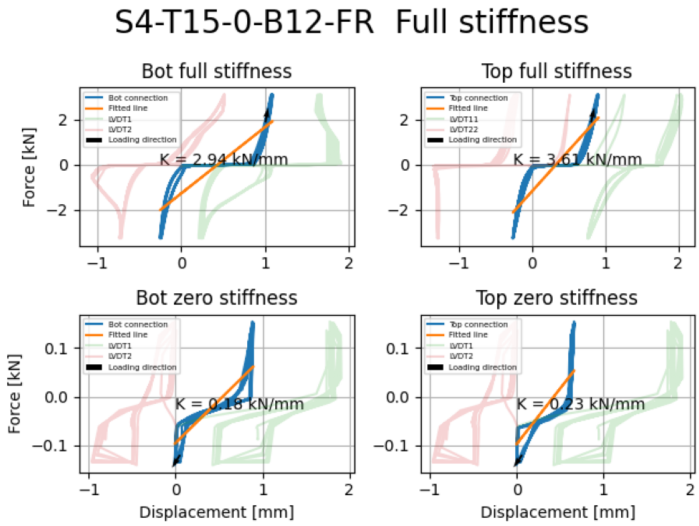

S2-T22-0-B1-Lower-FR
Bot full stiffness
K = 3.57 kN/mm
Top full stiffness
K = 6.04 kN/mm
Bot zero stiffness
K = 0.21 kN/mm
Top zero stiffness
K = 0.53 kN/mm
Force [kN]
Displacement [mm]

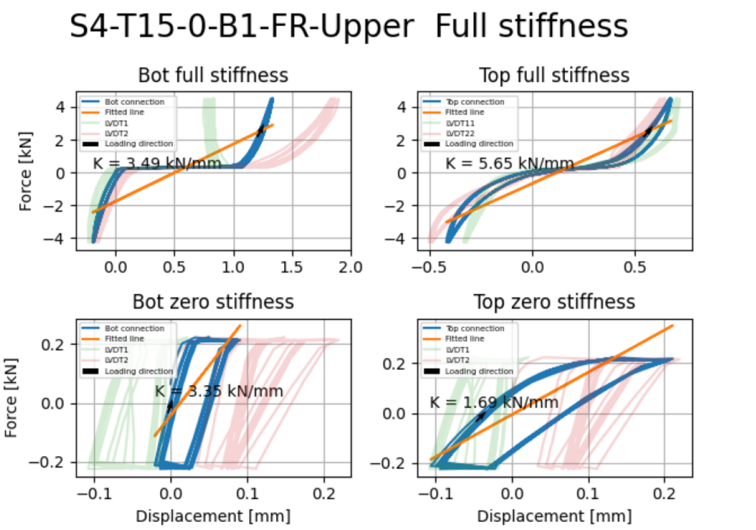

S2-T22-0-B12-FR
Bot full stiffness
K = 6.16 kN/mm
Top full stiffness
K = 2.94 kN/mm
Bot zero stiffness
K = 0.93 kN/mm
Top zero stiffness
K = 0.19 kN/mm
Force [kN]
Displacement [mm]

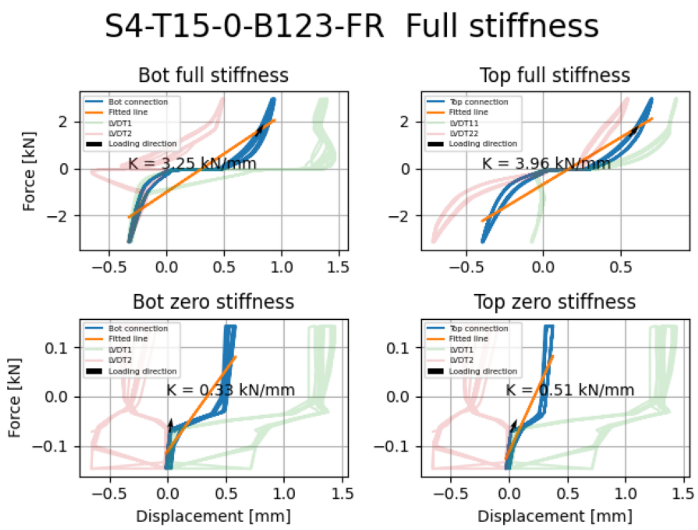

S2-T22-0-B1-Upper-FR
Bot full stiffness
K = 3.19 kN/mm
Top full stiffness
K = 11.35 kN/mm
Bot zero stiffness
K = 0.20 kN/mm
Top zero stiffness
K = 1.30 kN/mm
Force [kN]
Displacement [mm]

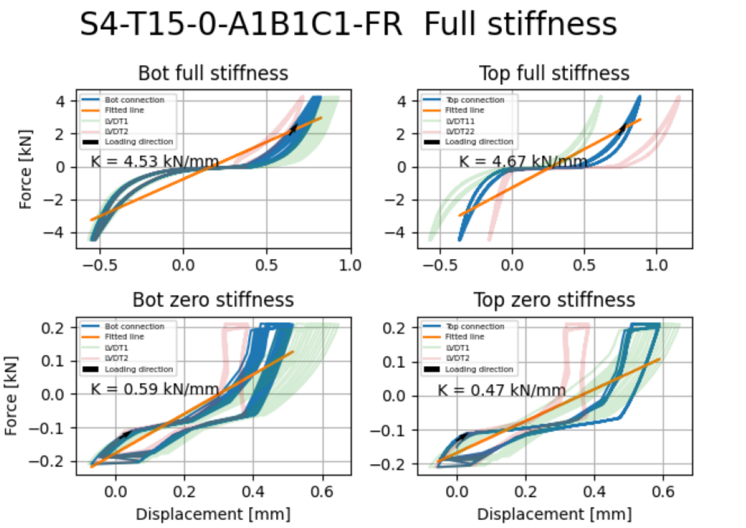

S2-T22-0-B123-FR
Bot full stiffness
K = 4.59 kN/mm
Top full stiffness
K = 3.17 kN/mm
Bot zero stiffness
K = 0.37 kN/mm
Top zero stiffness
K = 0.29 kN/mm
Force [kN]
Displacement [mm]

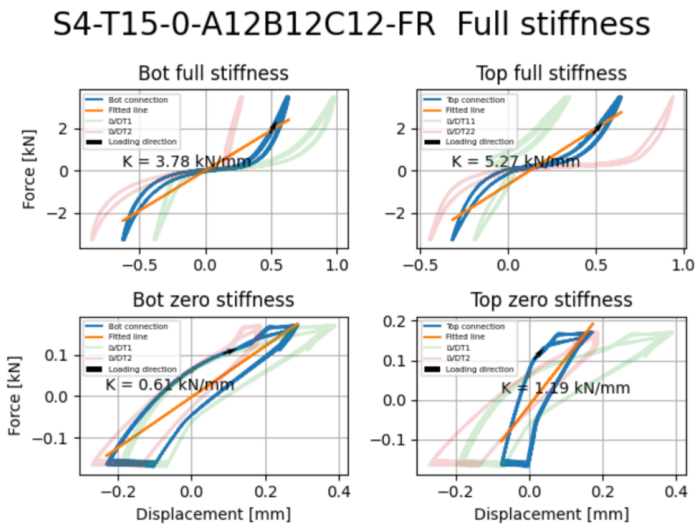

S4-T15-0-B1-FR-Lower Full stiffness
Bot full stiffness
Top full stiffness
K = 3.11 kN/mm
K = 4.93 kN/mm
Bot zero stiffness
Top zero stiffness
K = 2.32 kN/mm
K = 1.49 kN/mm
Force [kN]
Displacement [mm]

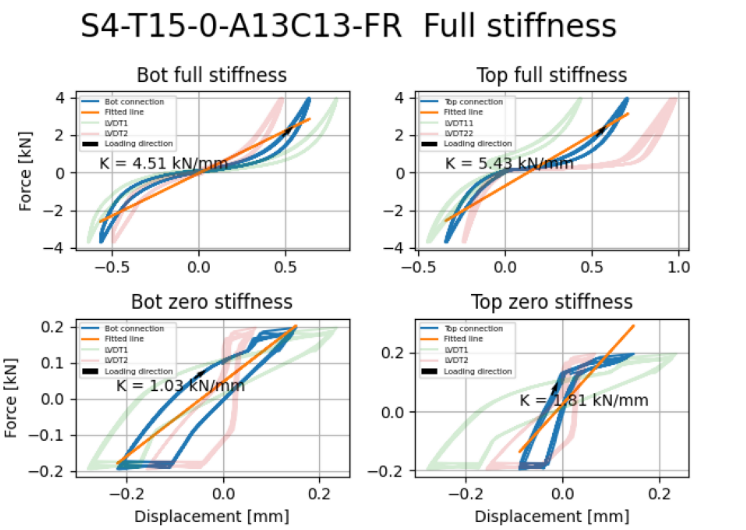

S4-T15-0-B12-FR Full stiffness
Bot full stiffness
Top full stiffness
K = 2.94 kN/mm
K = 3.61 kN/mm
Bot zero stiffness
Top zero stiffness
K = 0.18 kN/mm
K = 0.23 kN/mm
Force [kN]
Displacement [mm]

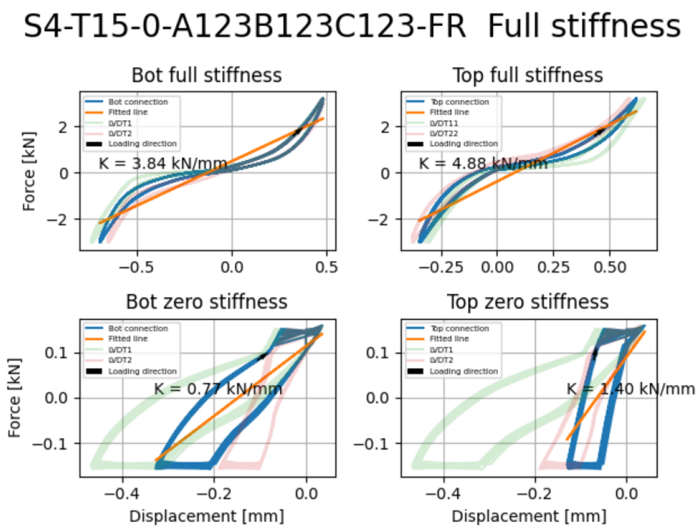

S4-T15-0-B1-FR-Upper Full stiffness
Bot full stiffness
Top full stiffness
K = 3.49 kN/mm
K = 5.65 kN/mm
Bot zero stiffness
Top zero stiffness
K = 3.35 kN/mm
K = 1.69 kN/mm
Force [kN]
Displacement [mm]

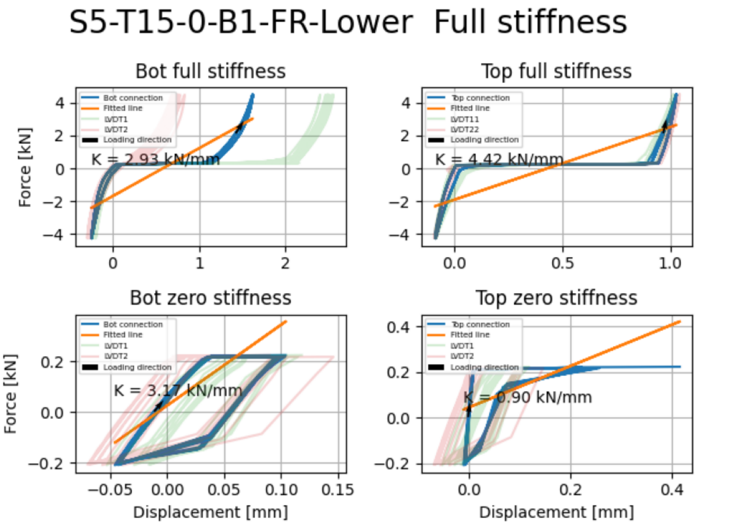

S4-T15-0-B123-FR Full stiffness
Bot full stiffness
Top full stiffness
K = 3.25 kN/mm
K = 3.96 kN/mm
Bot zero stiffness
Top zero stiffness
K = 0.33 kN/mm
K = 0.51 kN/mm
Force [kN]
Displacement [mm]

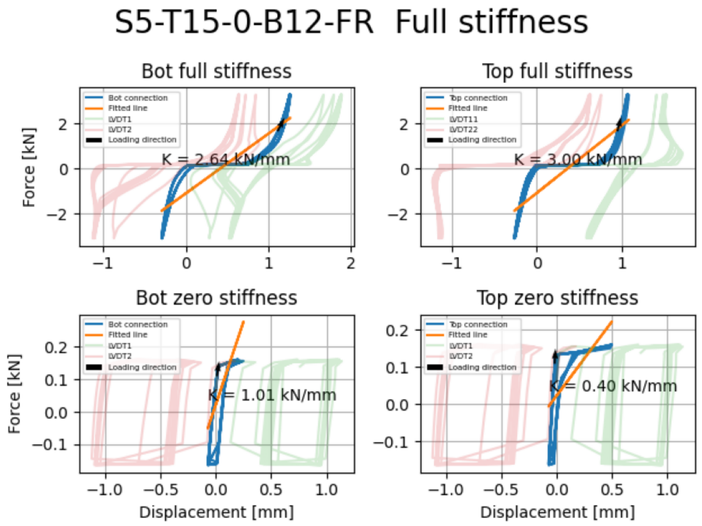

S4-T15-0-A1B1C1-FR Full stiffness
Bot full stiffness
Top full stiffness
Bot zero stiffness
Top zero stiffness
K = 4.53 kN/mm
K = 4.67 kN/mm
K = 0.59 kN/mm
K = 0.47 kN/mm
Force [kN]
Displacement [mm]

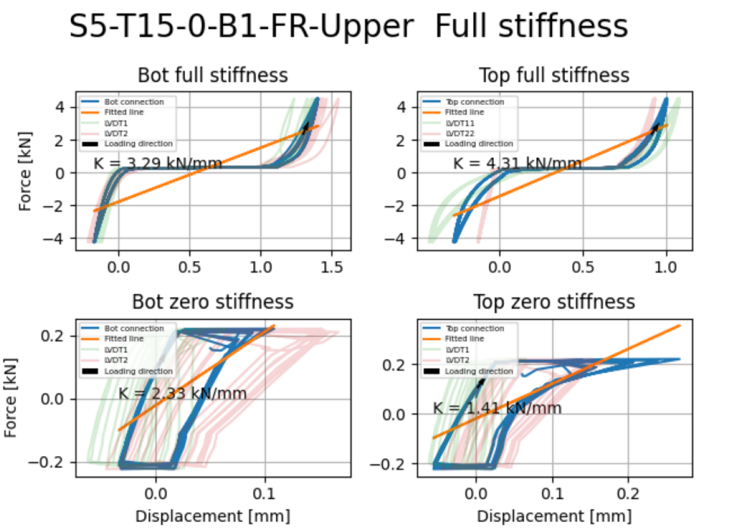

S4-T15-0-A12B12C12-FR Full stiffness
Bot full stiffness
Top full stiffness
Bot zero stiffness
Top zero stiffness
K = 3.78 kN/mm
K = 5.27 kN/mm
K = 0.61 kN/mm
K = 1.19 kN/mm
Force [kN]
Displacement [mm]

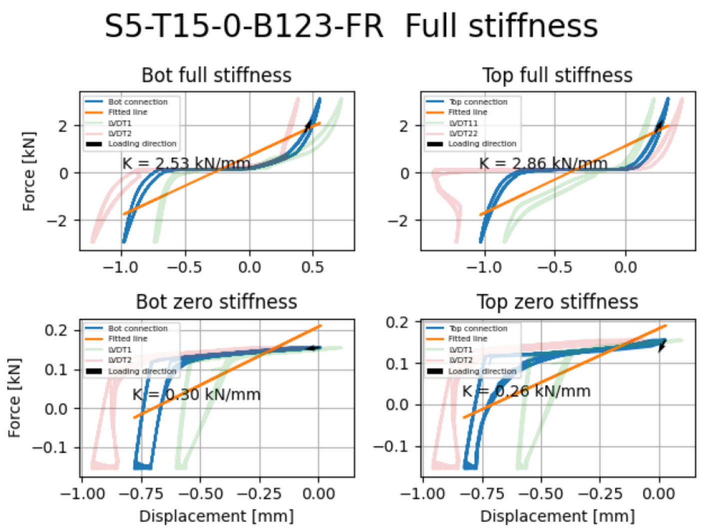

S4-T15-0-A13C13-FR Full stiffness
Bot full stiffness
Top full stiffness
Bot zero stiffness
Top zero stiffness
K = 4.51 kN/mm
K = 5.43 kN/mm
K = 1.03 kN/mm
K = 1.81 kN/mm
Force [kN]
Displacement [mm]

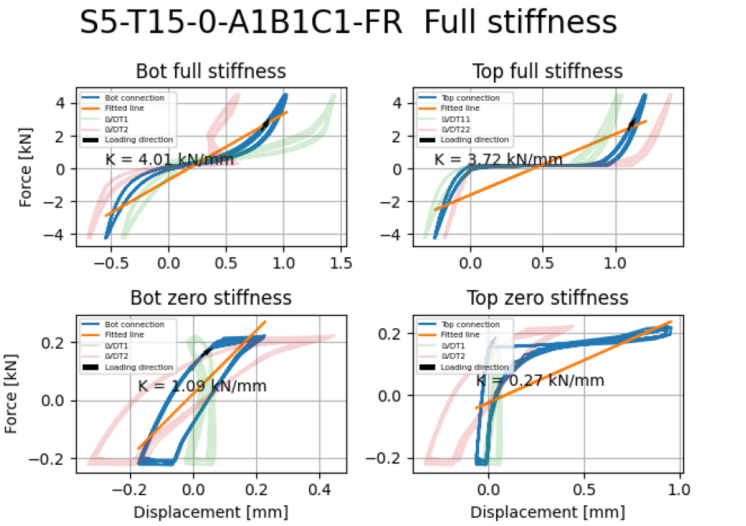

S4-T15-0-A123B123C123-FR Full stiffness
Bot full stiffness
Top full stiffness
Bot zero stiffness
Top zero stiffness
K = 3.84 kN/mm
K = 4.88 kN/mm
K = 0.77 kN/mm
K = 1.40 kN/mm
Force [kN]
Displacement [mm]

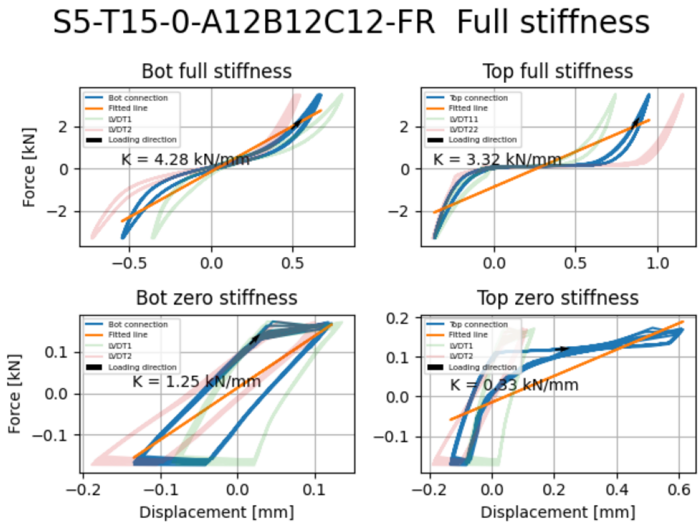

S5-T15-0-B1-FR-Lower Full stiffness
Bot full stiffness
Top full stiffness
Bot zero stiffness
Top zero stiffness
K = 2.93 kN/mm
K = 4.42 kN/mm
K = 3.17 kN/mm
K = 0.90 kN/mm
Force [kN]
Displacement [mm]

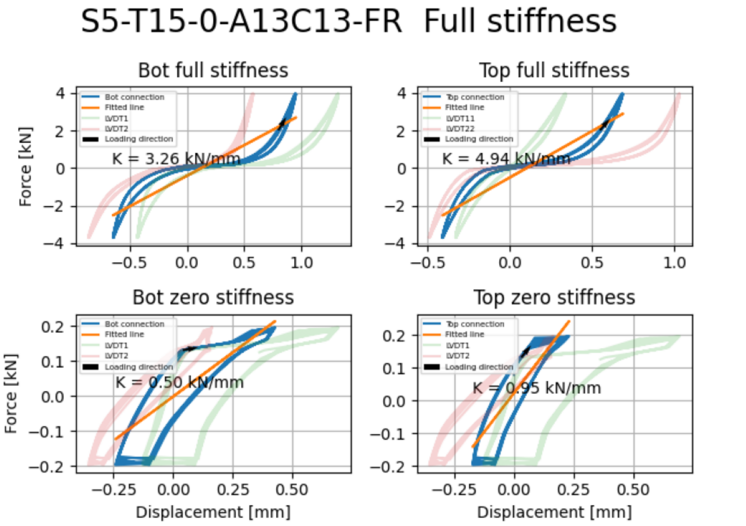

S5-T15-0-B12-FR Full stiffness
Bot full stiffness
Top full stiffness
Bot zero stiffness
Top zero stiffness
K = 2.64 kN/mm
K = 3.00 kN/mm
K = 1.01 kN/mm
K = 0.40 kN/mm
Force [kN]
Displacement [mm]

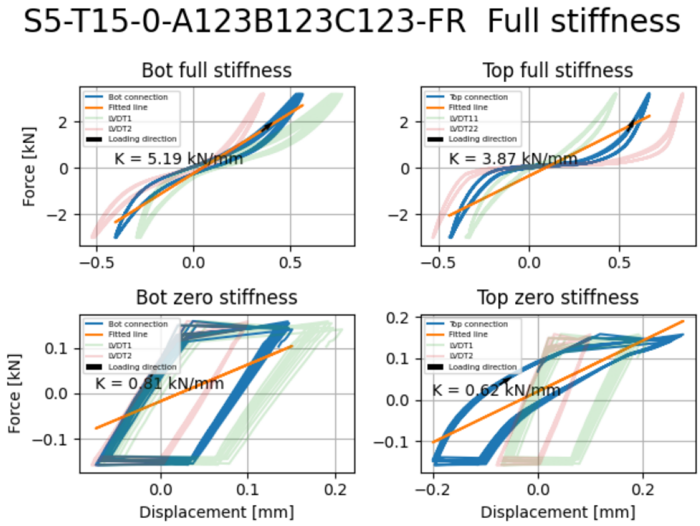

S5-T15-0-B1-FR-Upper Full stiffness
Bot full stiffness
Top full stiffness
Bot zero stiffness
Top zero stiffness
K = 3.29 kN/mm
K = 4.31 kN/mm
K = 2.33 kN/mm
K = 1.41 kN/mm
Force [kN]
Displacement [mm]

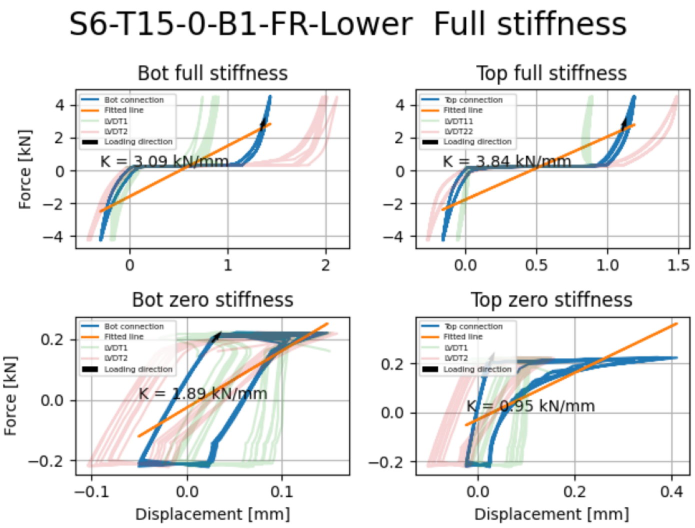

S5-T15-0-B123-FR Full stiffness
Bot full stiffness
Top full stiffness
Bot zero stiffness
Top zero stiffness
K = 2.53 kN/mm
K = 2.86 kN/mm
K = 0.30 kN/mm
K = 0.26 kN/mm
Force [kN]
Displacement [mm]

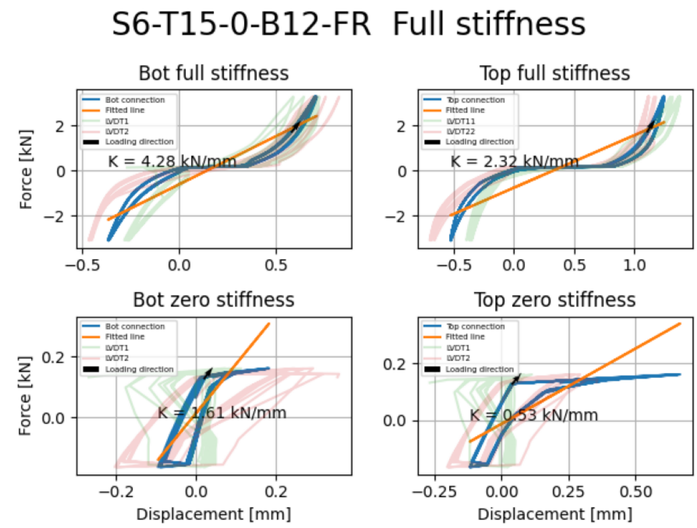

S5-T15-0-A1B1C1-FR Full stiffness
Bot full stiffness
K = 4.01 kN/mm
Top full stiffness
K = 3.72 kN/mm
Bot zero stiffness
K = 1.09 kN/mm
Top zero stiffness
K = 0.27 kN/mm
Force [kN]
Displacement [mm]

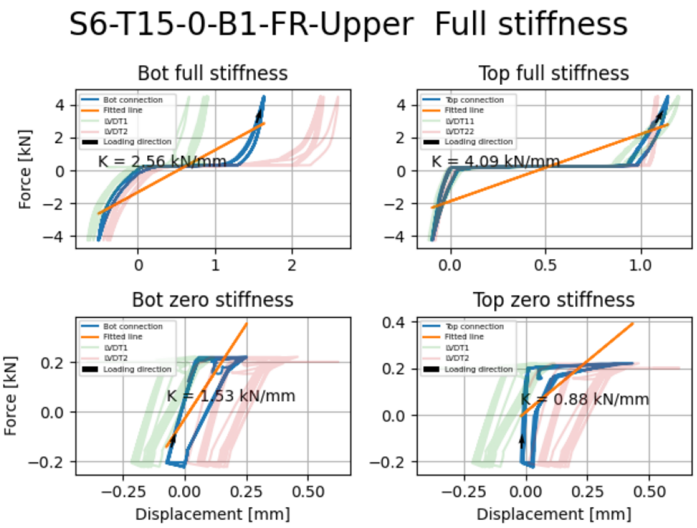

S5-T15-0-A12B12C12-FR Full stiffness
Bot full stiffness
K = 4.28 kN/mm
Top full stiffness
K = 3.32 kN/mm
Bot zero stiffness
K = 1.25 kN/mm
Top zero stiffness
K = 0.33 kN/mm
Force [kN]
Displacement [mm]

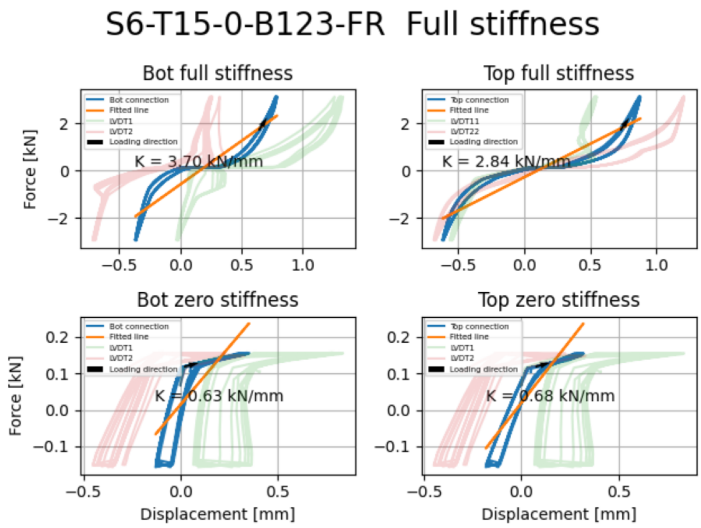

S5-T15-0-A13C13-FR Full stiffness
Bot full stiffness
K = 3.26 kN/mm
Top full stiffness
K = 4.94 kN/mm
Bot zero stiffness
K = 0.50 kN/mm
Top zero stiffness
K = 0.95 kN/mm
Force [kN]
Displacement [mm]

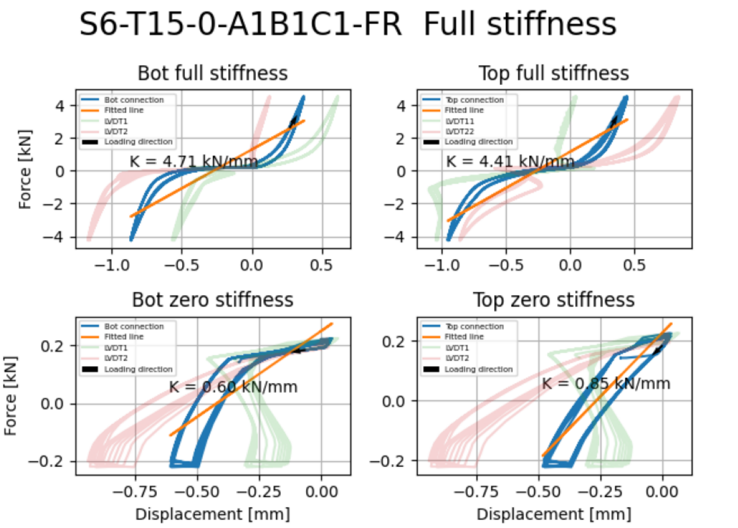

S5-T15-0-A123B123C123-FR Full stiffness
Bot full stiffness
K = 5.19 kN/mm
Top full stiffness
K = 3.87 kN/mm
Bot zero stiffness
K = 0.81 kN/mm
Top zero stiffness
K = 0.62 kN/mm
Force [kN]
Displacement [mm]


S6-T15-0-B1-FR-Lower Full stiffness
Bot full stiffness
K = 3.09 kN/mm
Top full stiffness
K = 3.84 kN/mm
Bot zero stiffness
K = 1.89 kN/mm
Top zero stiffness
K = 0.95 kN/mm
Force [kN]
Displacement [mm]

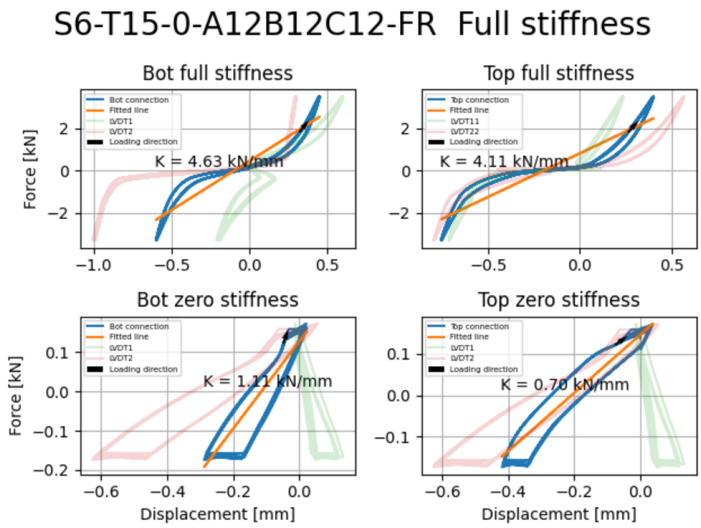

S6-T15-0-B12-FR Full stiffness
Bot full stiffness
K = 4.28 kN/mm
Top full stiffness
K = 2.32 kN/mm
Bot zero stiffness
K = 1.61 kN/mm
Top zero stiffness
K = 0.53 kN/mm
Force [kN]
Displacement [mm]


S6-T15-0-B1-FR-Upper Full stiffness
Bot full stiffness
K = 2.56 kN/mm
Top full stiffness
K = 4.09 kN/mm
Bot zero stiffness
K = 1.53 kN/mm
Top zero stiffness
K = 0.88 kN/mm
Force [kN]
Displacement [mm]

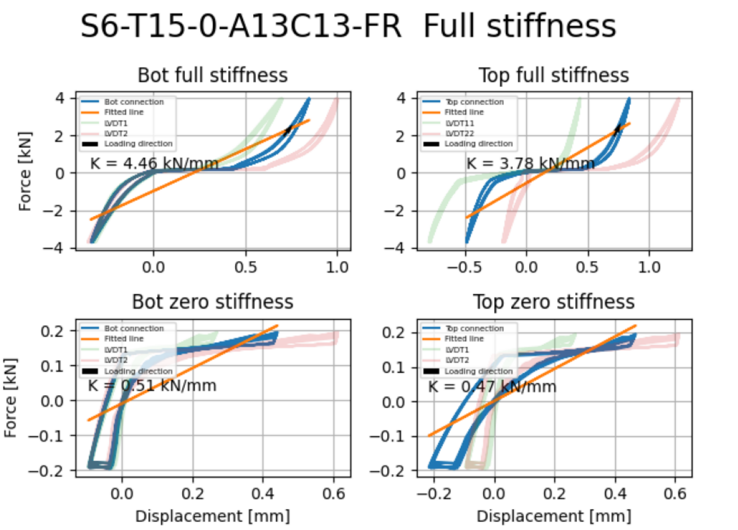

S6-T15-0-B123-FR Full stiffness
Bot full stiffness
K = 3.70 kN/mm
Top full stiffness
K = 2.84 kN/mm
Bot zero stiffness
K = 0.63 kN/mm
Top zero stiffness
K = 0.68 kN/mm
Force [kN]
Displacement [mm]



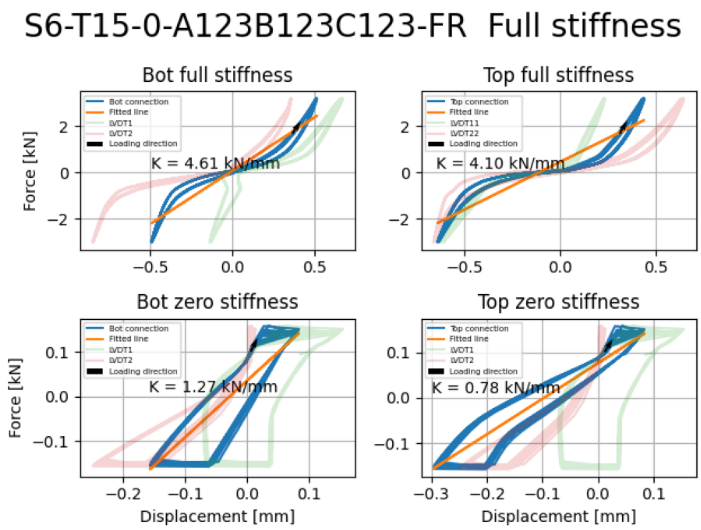
S6-T15-0-A1B1C1-FR Full stiffness
Bot full stiffness
Top full stiffness
K = 4.71 kN/mm
K = 4.41 kN/mm
Bot zero stiffness
Top zero stiffness
K = 0.60 kN/mm
K = 0.85 kN/mm
Force [kN]
Displacement [mm]



S6-T15-0-A12B12C12-FR Full stiffness
Bot full stiffness
Top full stiffness
K = 4.63 kN/mm
K = 4.11 kN/mm
Bot zero stiffness
Top zero stiffness
K = 1.11 kN/mm
K = 0.70 kN/mm
Force [kN]
Displacement [mm]

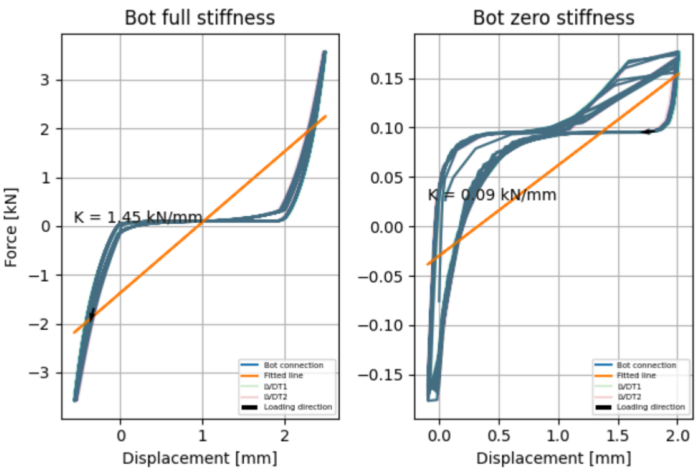


S6-T15-0-A13C13-FR Full stiffness
Bot full stiffness
Top full stiffness
K = 4.46 kN/mm
K = 3.78 kN/mm
Bot zero stiffness
Top zero stiffness
K = 0.51 kN/mm
K = 0.47 kN/mm
Force [kN]
Displacement [mm]

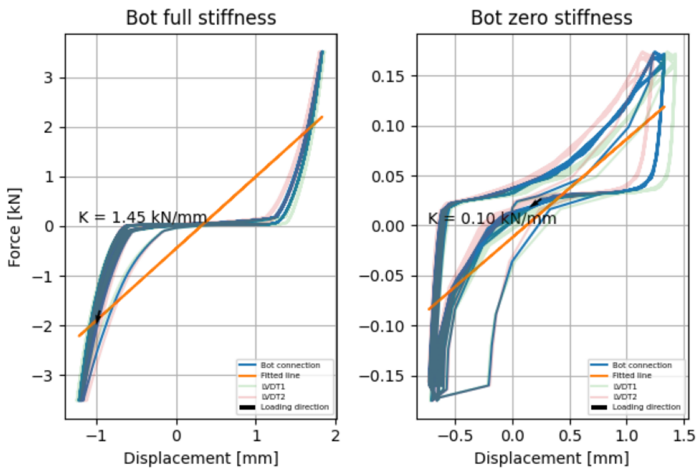


S6-T15-0-A123B123C123-FR Full stiffness
Bot full stiffness
Top full stiffness
K = 4.61 kN/mm
K = 4.10 kN/mm
Bot zero stiffness
Top zero stiffness
K = 1.27 kN/mm
K = 0.78 kN/mm
Force [kN]
Displacement [mm]

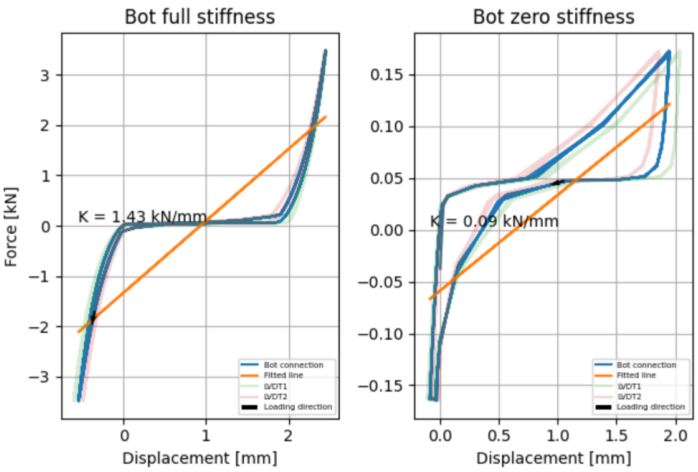
S7-T15-90-B2-FR



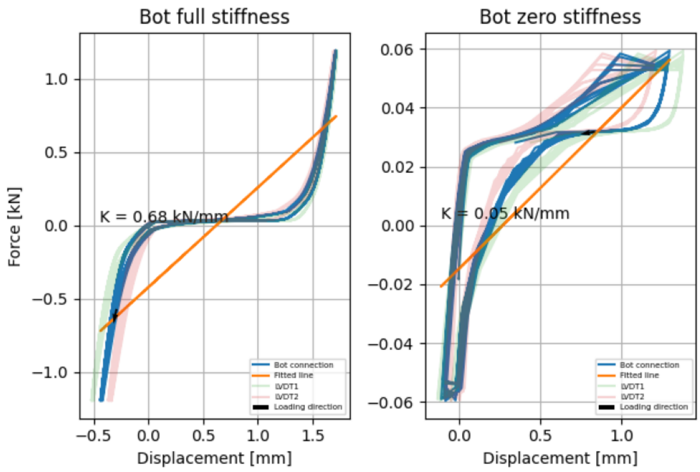
S7-T15-90-A2B2C2-FR



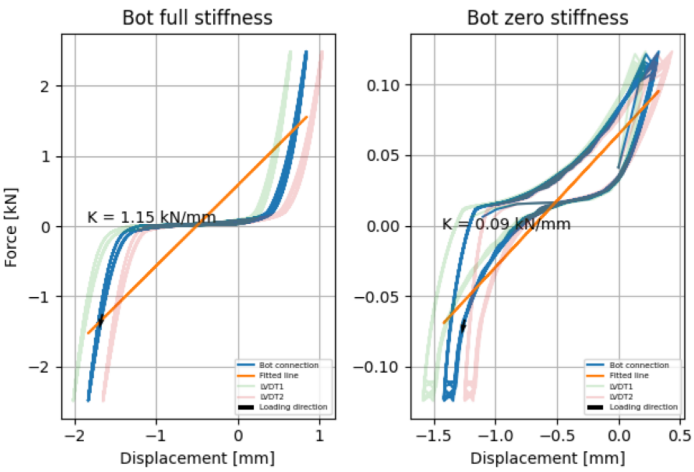
S7-T15-90-A2C2-FR



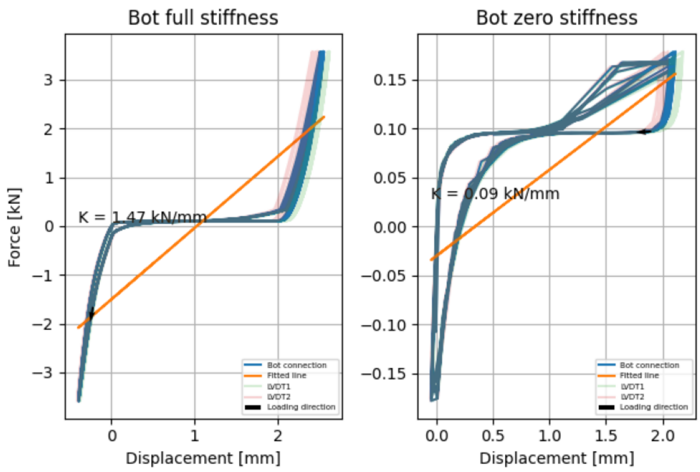
S7-T15-90-B123-FR



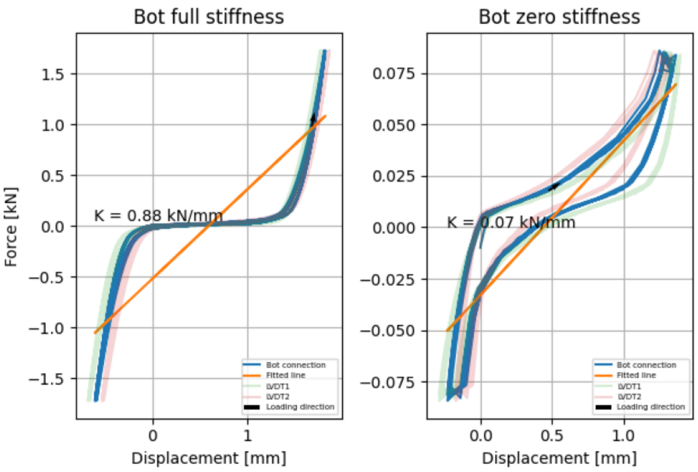
S7-T15-90-A123C123-FR



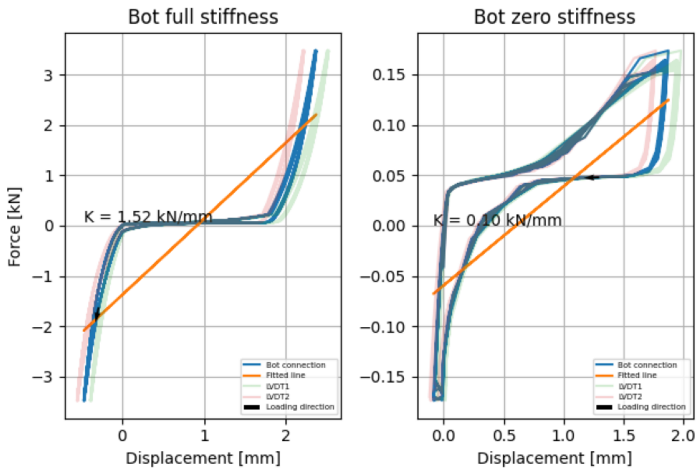
S8-T15-90-B2-FR

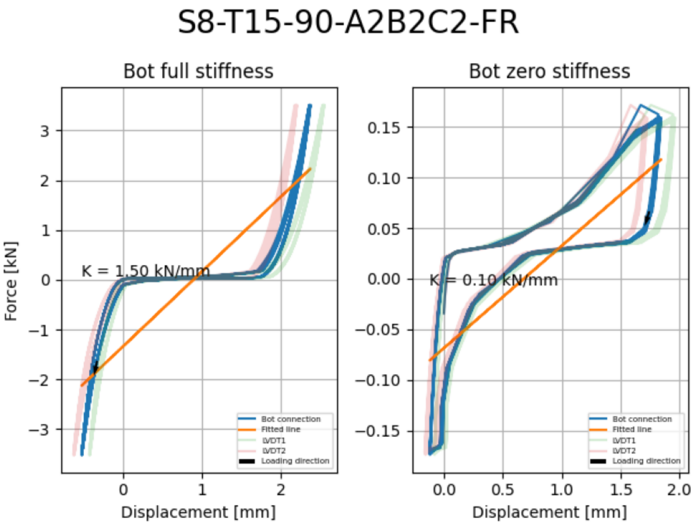


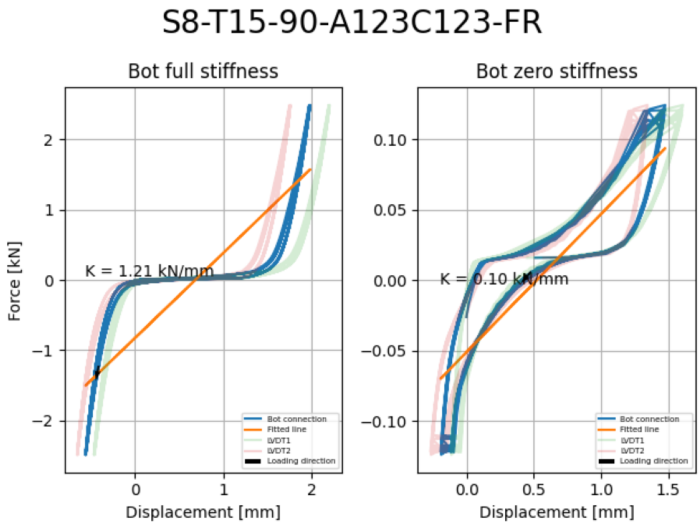
S7-T15-90-A123B123C123-FR

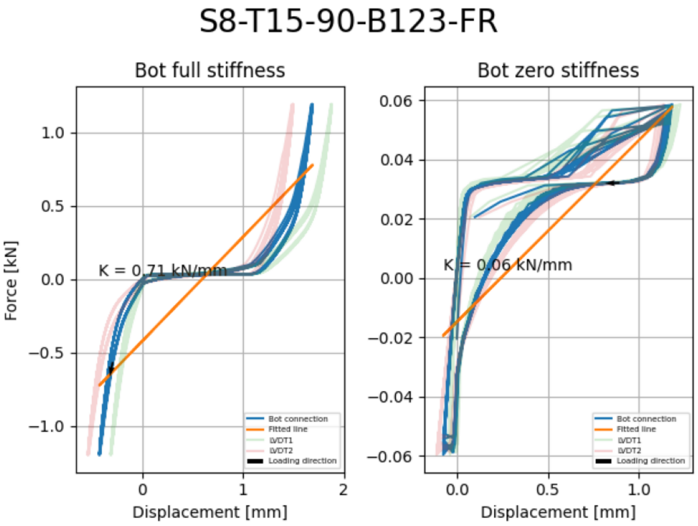


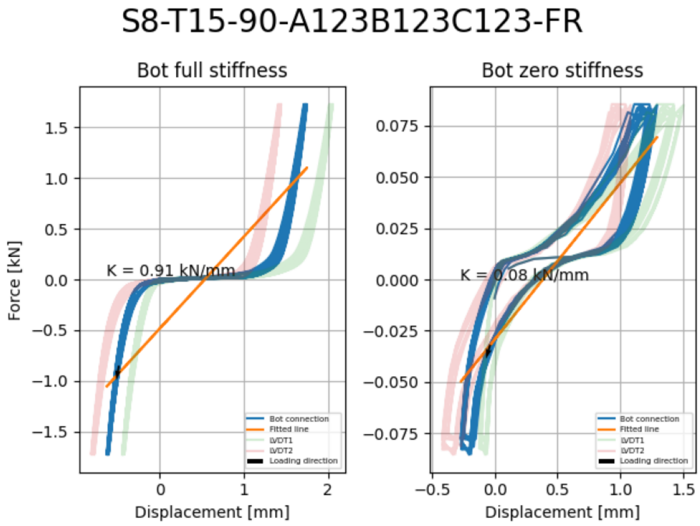
S8-T15-90-A2C2-FR

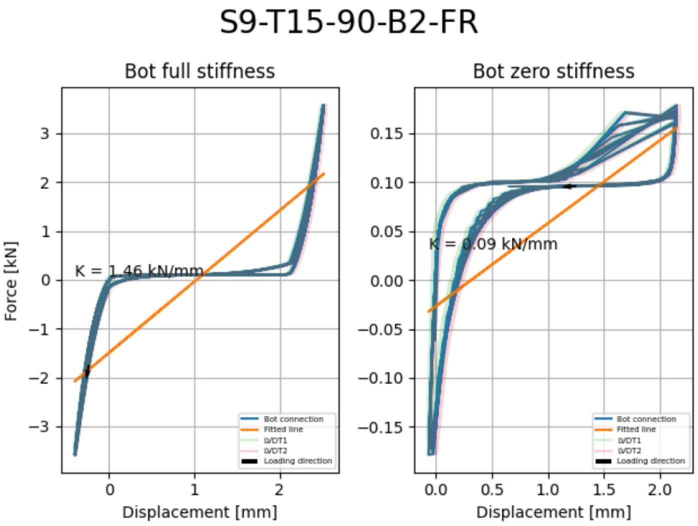


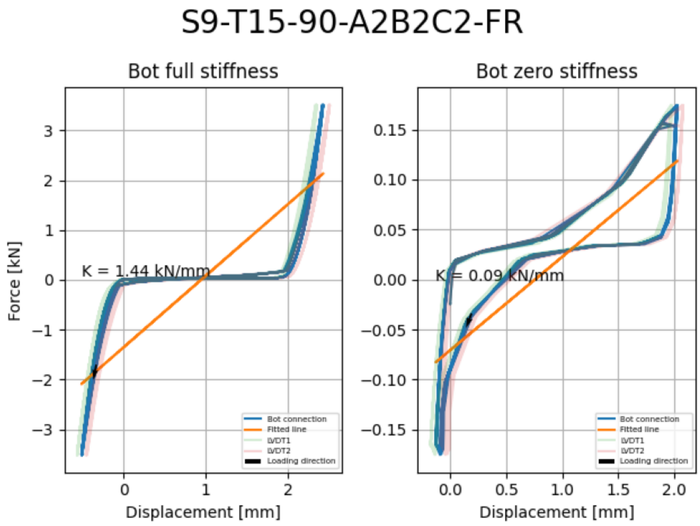

S8-T15-90-A2B2C2-FR
Bot full stiffness
Bot zero stiffness
K = 1.50 kN/mm
K = 0.10 kN/mm
Force [kN]
Displacement [mm]
Displacement [mm]
Bot connection
Fitted line
LVDT1
LVDT2
Loading direction

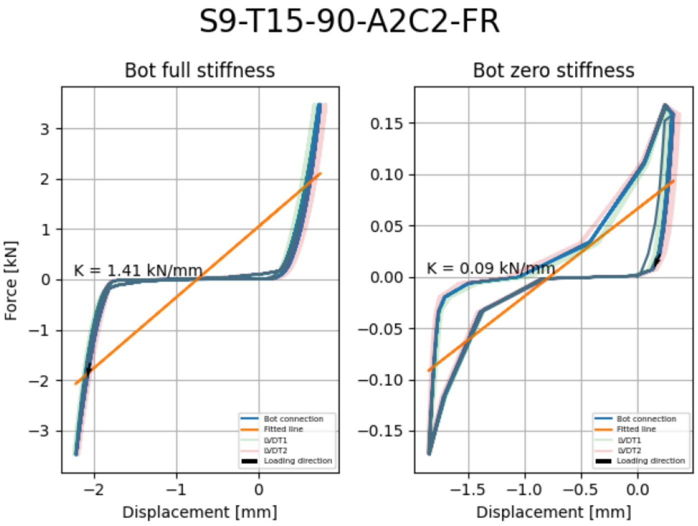

S8-T15-90-A123C123-FR
Bot full stiffness
Bot zero stiffness
K = 1.21 kN/mm
K = 0.10 kN/mm
Force [kN]
Displacement [mm]
Displacement [mm]
Bot connection
Fitted line
LVDT1
LVDT2
Loading direction

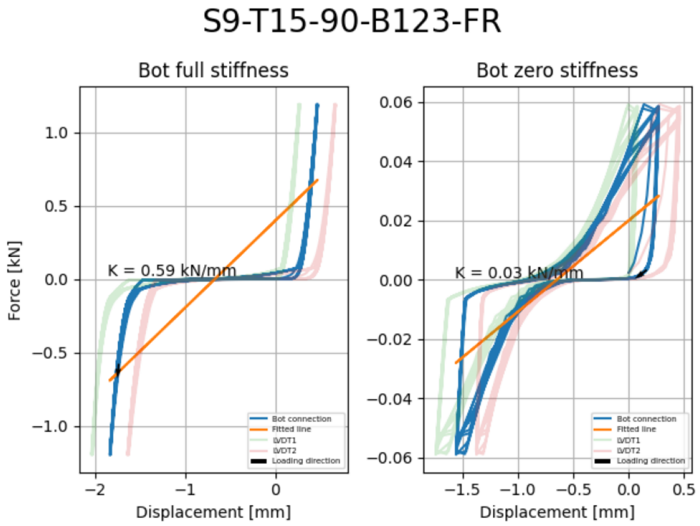

S8-T15-90-B123-FR
Bot full stiffness
Bot zero stiffness
K = 0.71 kN/mm
K = 0.06 kN/mm
Force [kN]
Displacement [mm]
Displacement [mm]
Bot connection
Fitted line
LVDT1
LVDT2
Loading direction


S8-T15-90-A123B123C123-FR
Bot full stiffness
Bot zero stiffness
K = 0.91 kN/mm
K = 0.08 kN/mm
Force [kN]
Displacement [mm]
Displacement [mm]
Bot connection
Fitted line
LVDT1
LVDT2
Loading direction


S9-T15-90-B2-FR
Bot full stiffness
K = 1.46 kN/mm
Bot zero stiffness
K = 0.09 kN/mm
Force [kN]
Displacement [mm]
Bot connection
Fitted line
LVDT1
LVDT2
Loading direction


S9-T15-90-A2B2C2-FR
Bot full stiffness
K = 1.44 kN/mm
Bot zero stiffness
K = 0.09 kN/mm
Force [kN]
Displacement [mm]
Bot connection
Fitted line
LVDT1
LVDT2
Loading direction


S9-T15-90-A2C2-FR
Bot full stiffness
K = 1.41 kN/mm
Bot zero stiffness
K = 0.09 kN/mm
Force [kN]
Displacement [mm]
Bot connection
Fitted line
LVDT1
LVDT2
Loading direction


S9-T15-90-B123-FR
Bot full stiffness
K = 0.59 kN/mm
Bot zero stiffness
K = 0.03 kN/mm
Force [kN]
Displacement [mm]
Bot connection
Fitted line
LVDT1
LVDT2
Loading direction

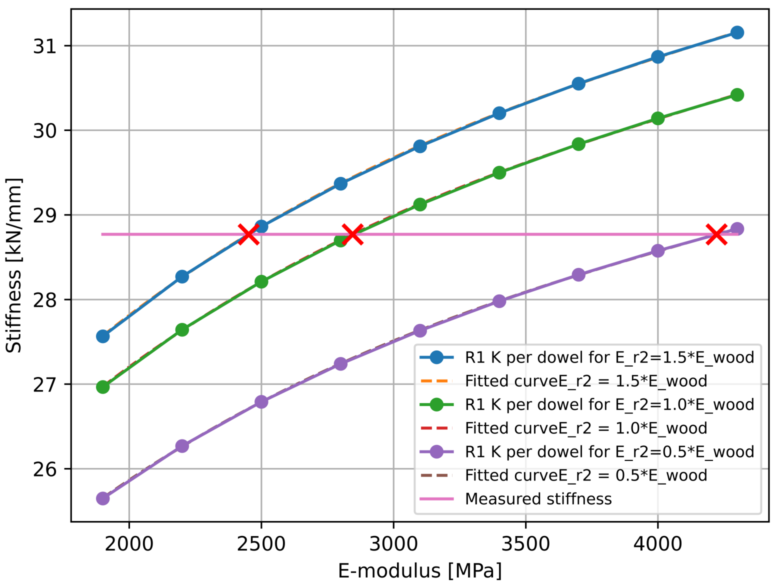
D Abaqus numerical results

This appendix includes all the numerical results (plots) obtained by analyses through Abaqus. For an complete overview, see table 4

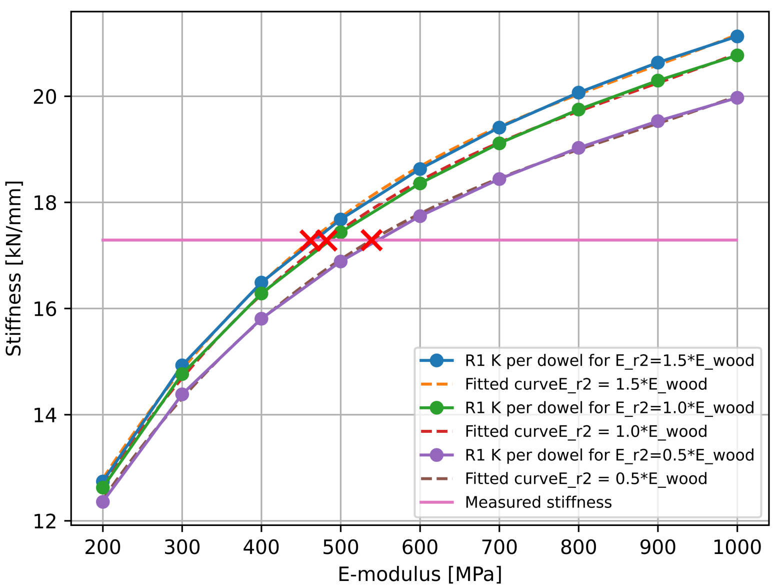
Table 4: Included documents in appendix D

Name	**Description**
Configuration S1-Configuration	Numerical results for analyses ran with specimen type 1 geometry. Three different configurations were ran, with corresponding notation system as used earlier. The measured stiffness is taken from Frette et al. (2021)
Configuration S2-Configuration	Numerical results for analyses ran with specimen type 1 geometry. Six different configurations were ran, with corresponding notation system as used earlier. The measured stiffness is taken from Frette et al. (2021)
Configuration S3-Configuration	Numerical results for analyses ran with specimen type 3 geometry. Four different configurations were ran, with corresponding notation system as used earlier. The measured stiffness is taken from Frette et al. (2021)

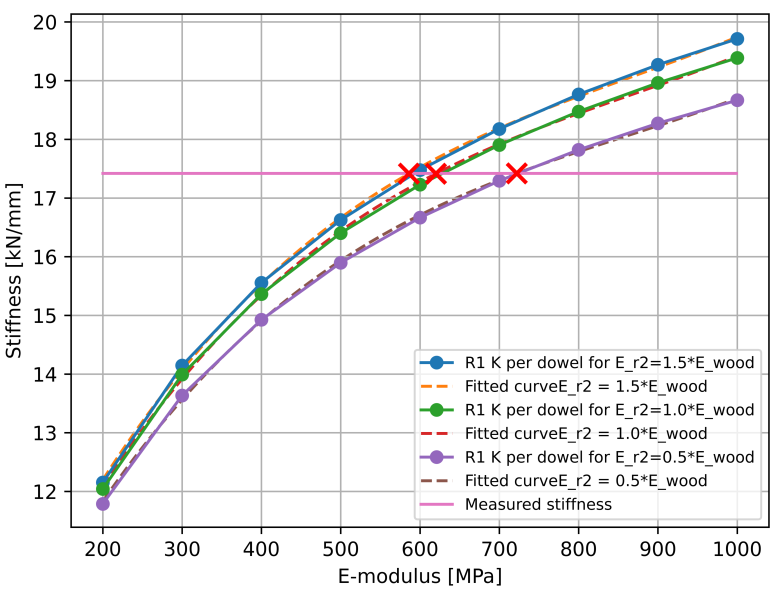
Configuration S1-B1

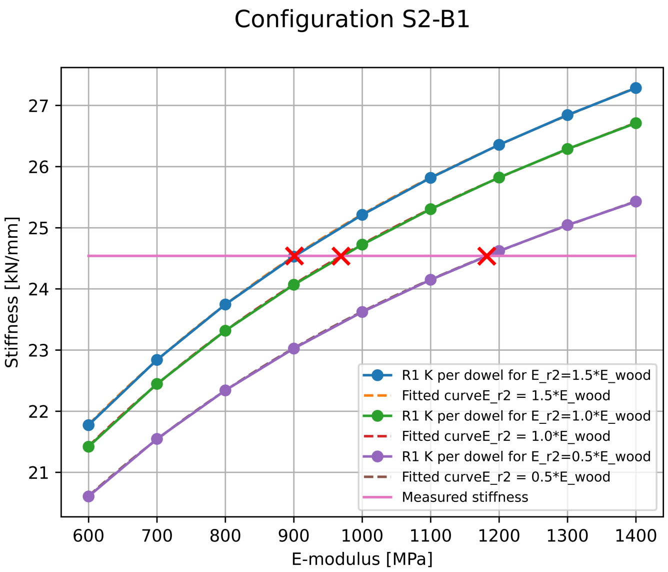


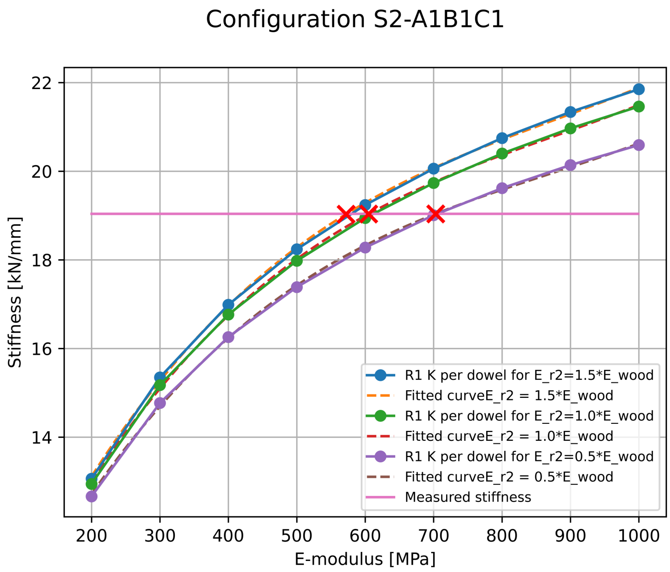
Configuration S1-B12

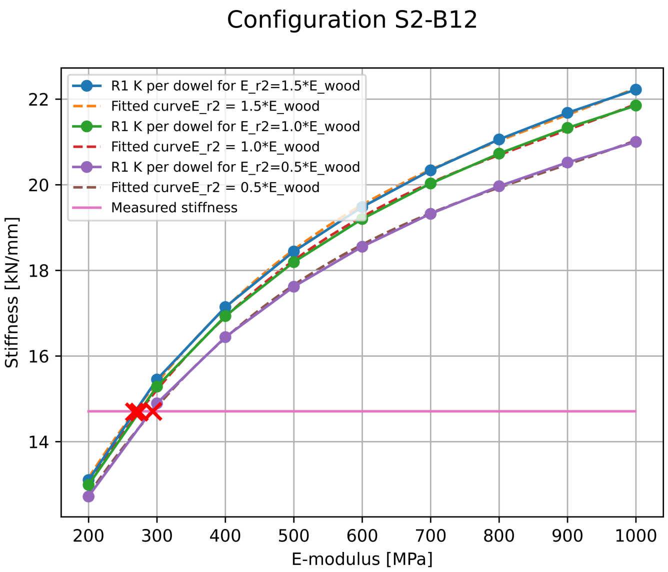


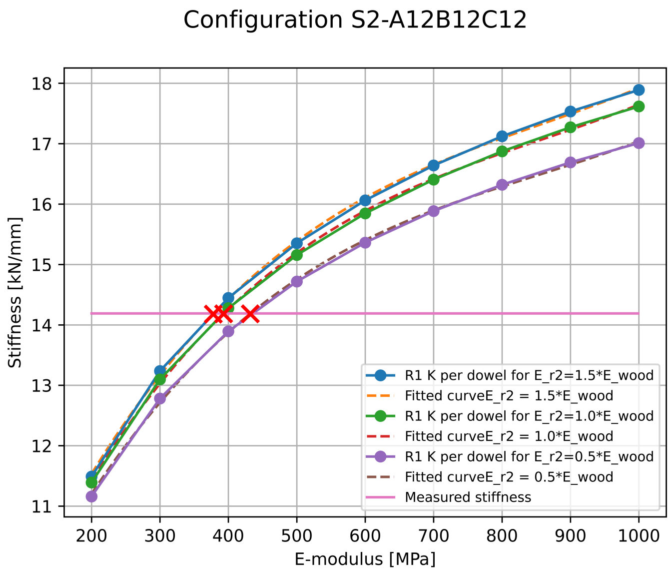
Configuration S1-B123

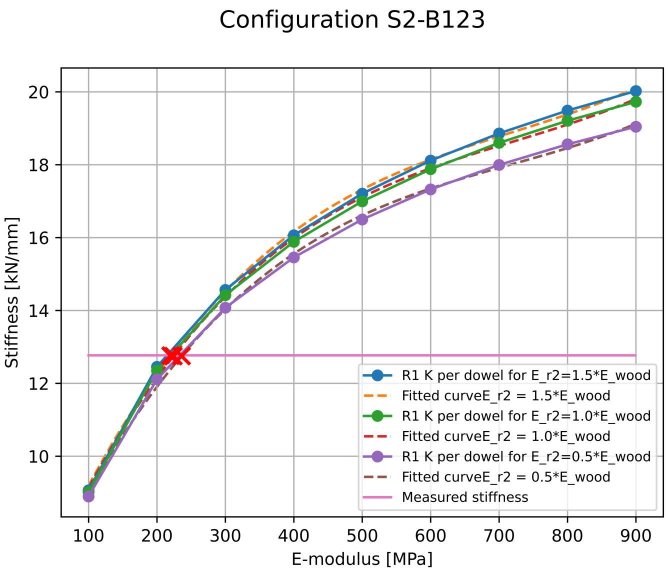


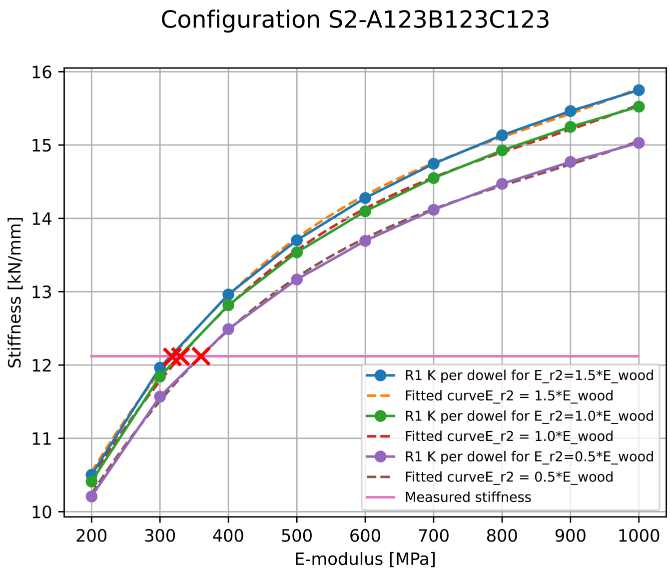

Configuration S2-B1
Stiffness [kN/mm]
E-modulus [MPa]
R1 K per dowel for E_r2=1.5*E_wood
Fitted curveE_r2 = 1.5*E_wood
R1 K per dowel for E_r2=1.0*E_wood
Fitted curveE_r2 = 1.0*E_wood
R1 K per dowel for E_r2=0.5*E_wood
Fitted curveE_r2 = 0.5*E_wood
Measured stiffness

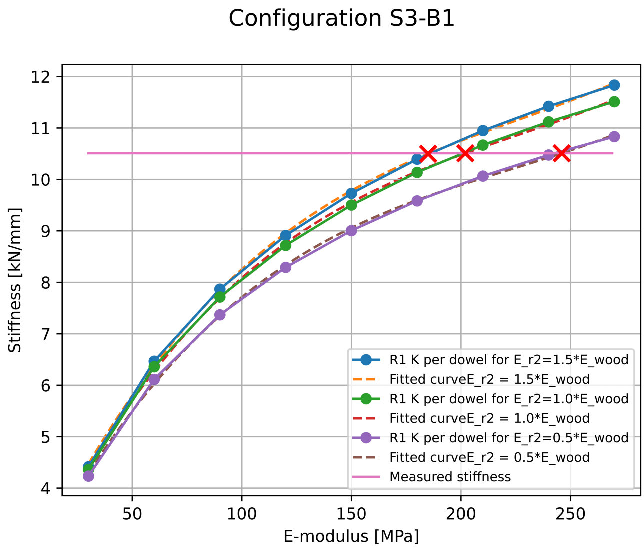

Configuration S2-A1B1C1
Stiffness [kN/mm]
E-modulus [MPa]
R1 K per dowel for E_r2=1.5*E_wood
Fitted curveE_r2 = 1.5*E_wood
R1 K per dowel for E_r2=1.0*E_wood
Fitted curveE_r2 = 1.0*E_wood
R1 K per dowel for E_r2=0.5*E_wood
Fitted curveE_r2 = 0.5*E_wood
Measured stiffness

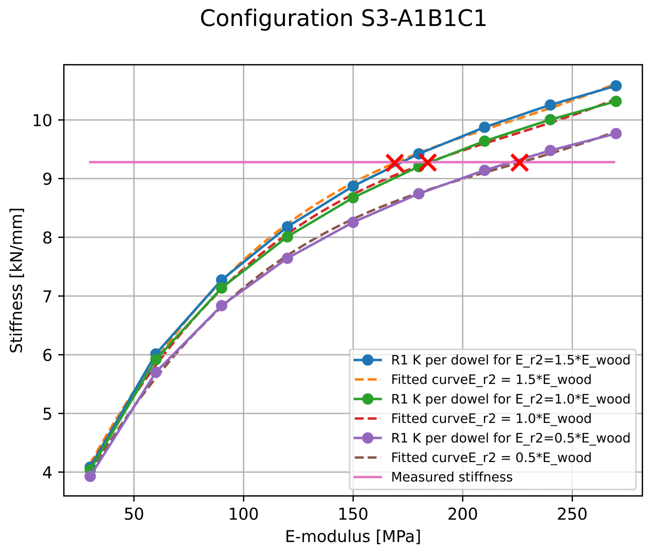

Configuration S2-B12
Stiffness [kN/mm]
E-modulus [MPa]
R1 K per dowel for E_r2=1.5*E_wood
Fitted curveE_r2 = 1.5*E_wood
R1 K per dowel for E_r2=1.0*E_wood
Fitted curveE_r2 = 1.0*E_wood
R1 K per dowel for E_r2=0.5*E_wood
Fitted curveE_r2 = 0.5*E_wood
Measured stiffness

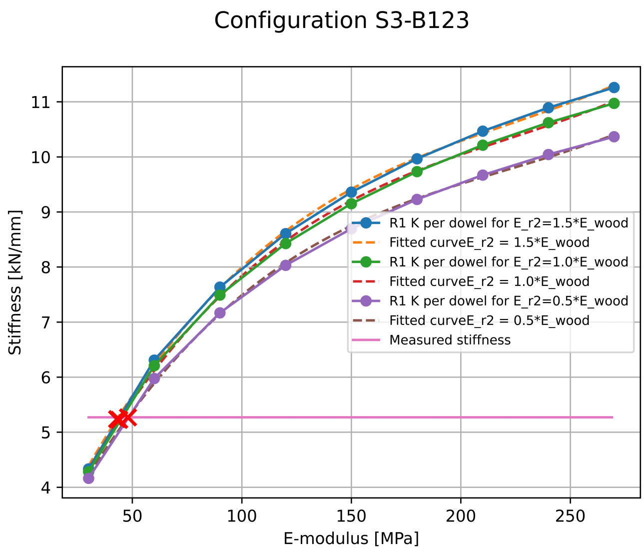

Configuration S2-A12B12C12
Stiffness [kN/mm]
E-modulus [MPa]
R1 K per dowel for E_r2=1.5*E_wood
Fitted curveE_r2 = 1.5*E_wood
R1 K per dowel for E_r2=1.0*E_wood
Fitted curveE_r2 = 1.0*E_wood
R1 K per dowel for E_r2=0.5*E_wood
Fitted curveE_r2 = 0.5*E_wood
Measured stiffness

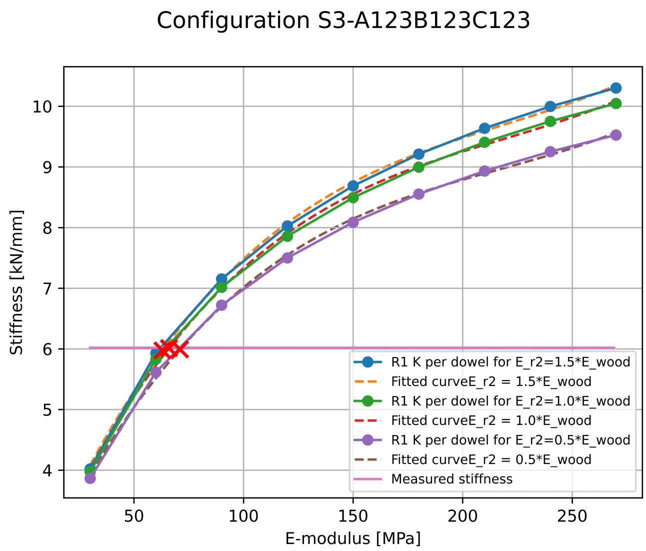

Configuration S2-B123
Stiffness [kN/mm]
E-modulus [MPa]
R1 K per dowel for E_r2=1.5*E_wood
Fitted curveE_r2 = 1.5*E_wood
R1 K per dowel for E_r2=1.0*E_wood
Fitted curveE_r2 = 1.0*E_wood
R1 K per dowel for E_r2=0.5*E_wood
Fitted curveE_r2 = 0.5*E_wood
Measured stiffness


Configuration S2-A123B123C123
Stiffness [kN/mm]
E-modulus [MPa]
R1 K per dowel for E_r2=1.5*E_wood
Fitted curveE_r2 = 1.5*E_wood
R1 K per dowel for E_r2=1.0*E_wood
Fitted curveE_r2 = 1.0*E_wood
R1 K per dowel for E_r2=0.5*E_wood
Fitted curveE_r2 = 0.5*E_wood
Measured stiffness


Configuration S3-B1
Stiffness [kN/mm]
E-modulus [MPa]
R1 K per dowel for E_r2=1.5*E_wood
Fitted curveE_r2 = 1.5*E_wood
R1 K per dowel for E_r2=1.0*E_wood
Fitted curveE_r2 = 1.0*E_wood
R1 K per dowel for E_r2=0.5*E_wood
Fitted curveE_r2 = 0.5*E_wood
Measured stiffness


Configuration S3-A1B1C1
Stiffness [kN/mm]
E-modulus [MPa]
R1 K per dowel for E_r2=1.5*E_wood
Fitted curveE_r2 = 1.5*E_wood
R1 K per dowel for E_r2=1.0*E_wood
Fitted curveE_r2 = 1.0*E_wood
R1 K per dowel for E_r2=0.5*E_wood
Fitted curveE_r2 = 0.5*E_wood
Measured stiffness


Configuration S3-B123
Stiffness [kN/mm]
E-modulus [MPa]
R1 K per dowel for E_r2=1.5*E_wood
Fitted curveE_r2 = 1.5*E_wood
R1 K per dowel for E_r2=1.0*E_wood
Fitted curveE_r2 = 1.0*E_wood
R1 K per dowel for E_r2=0.5*E_wood
Fitted curveE_r2 = 0.5*E_wood
Measured stiffness

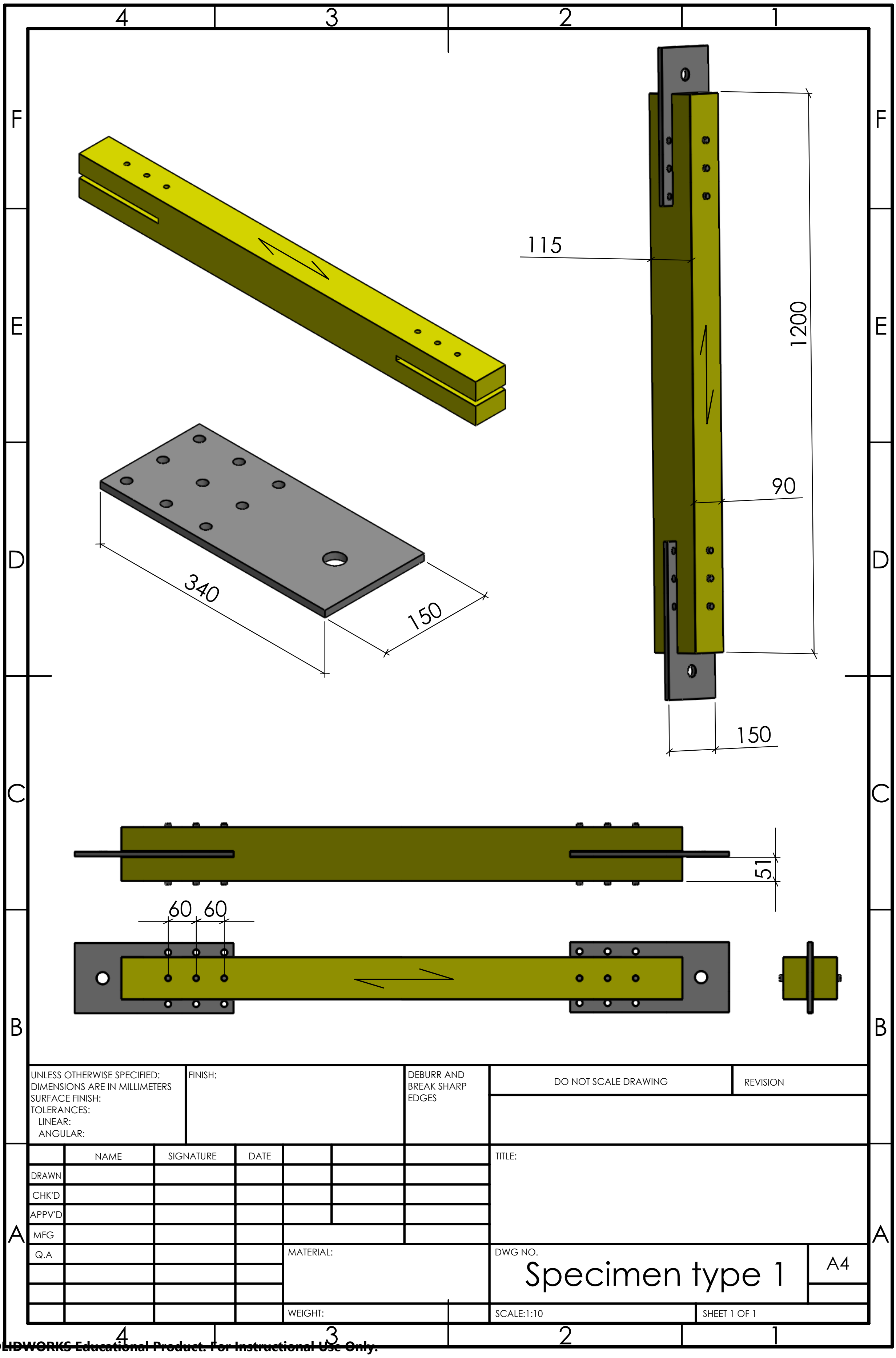

Configuration S3-A123B123C123
Stiffness [kN/mm]
E-modulus [MPa]
R1 K per dowel for E_r2=1.5*E_wood
Fitted curveE_r2 = 1.5*E_wood
R1 K per dowel for E_r2=1.0*E_wood
Fitted curveE_r2 = 1.0*E_wood
R1 K per dowel for E_r2=0.5*E_wood
Fitted curveE_r2 = 0.5*E_wood
Measured stiffness

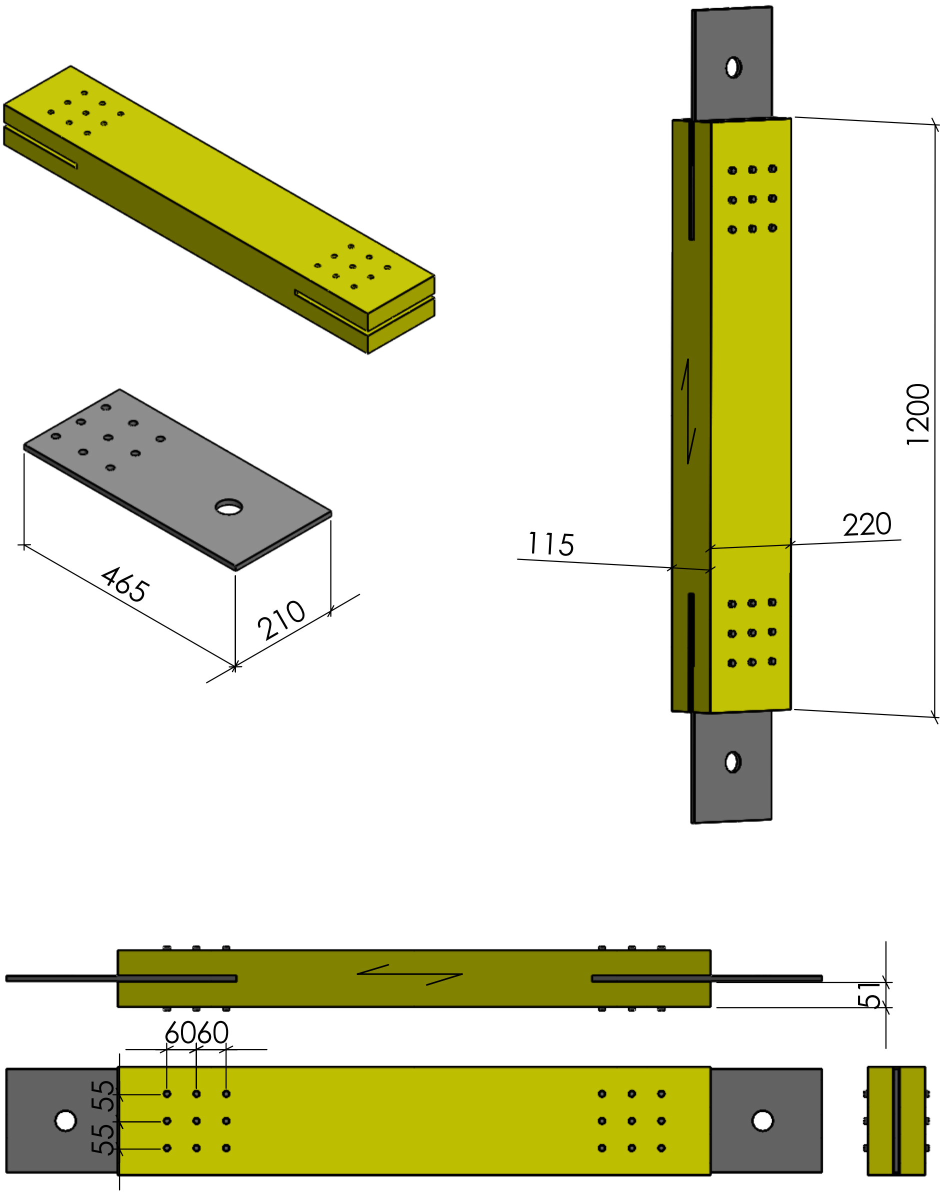
E Drawings

This appendix includes all drawings produced during the master work. The drawings are produced in Solidworks 2021. The seperate parts and assemblies are attached in the digital appendix, while the PDF-drawings are presented here, see table 5

Table 5: Included documents in appendix E

Name	Description
Specimen type 1	Dimensions of specimen type 1
Specimen type 2	Dimensions of specimen type 2
Specimen type 3	Dimensions of specimen type 3
Stålplate type 1 & 2	Dimensions of steel plate used by \textcite{frette_experimental_2021}. This steel plate was used when specimen type 1 was tested in the 100kN actuator, but as the cyclic test series were carried out all over again in the 400 kN actuator, this drawing could have been left out. It is, however, included for completeness.
Stålplate type 1 & 2 Stor plate	Dimensions of steel plate designed and used in 400 kN actuator.
Stålplate type 3	Dimensions of steel plate used for specimen type 3. The plate was designed and produced by frette.
Stålplate type 3 Failure test	Dimensions of modified steel plate type 3 used for failure test in the 250 kN actuator.
Type 3 failure test setup	Complete setup of the parts created in order to perform the failure test of specimen type 3 in the 250 kN actuator.


4
3
2
1
F
E
D
C
B
A
115
1200
90
150
340
150
51
60
60
UNLESS OTHERWISE SPECIFIED:
DIMENSIONS ARE IN MILLIMETERS
SURFACE FINISH:
TOLERANCES:
LINEAR:
ANGULAR:
FINISH:
DEBURR AND BREAK SHARP EDGES
DO NOT SCALE DRAWING
REVISION
NAME
SIGNATURE
DATE
DRAWN
CHK'D
APPV'D
MFG
Q.A
TITLE:
MATERIAL:
WEIGHT:
DWG NO.
Specimen type 1
A4
SCALE:1:10
SHEET 1 OF 1



UNLESS OTHERWISE SPECIFIED: DIMENSIONS ARE IN MILLIMETERS SURFACE FINISH: TOLERANCES: LINEAR: ANGULAR:	FINISH:	DEBURR AND BREAK SHARP EDGES	DO NOT SCALE DRAWING / REVISION

	NAME	SIGNATURE	DATE
DRAWN			
CHK'D			
APPV'D			
MFG			
Q.A			

TITLE:

MATERIAL:

WEIGHT:

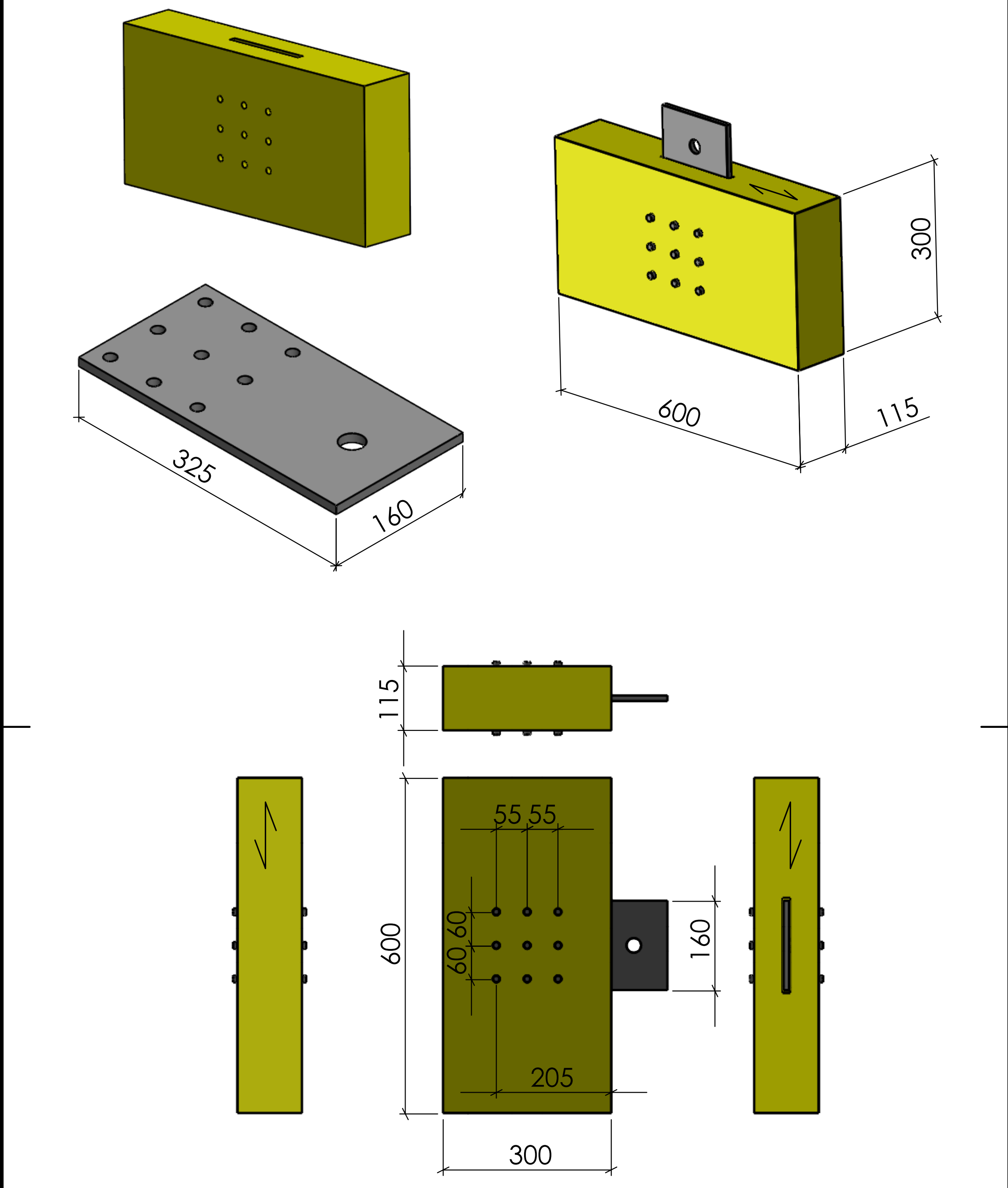
DWG NO.

Specimen type 2

A4

SCALE:1:15

SHEET 1 OF 1



UNLESS OTHERWISE SPECIFIED:
DIMENSIONS ARE IN MILLIMETERS
SURFACE FINISH:
TOLERANCES:
LINEAR:
ANGULAR:

FINISH:

DEBURR AND BREAK SHARP EDGES

DO NOT SCALE DRAWING

REVISION

	NAME	SIGNATURE	DATE		
DRAWN					
CHK'D					
APPV'D					
MFG					
Q.A					

TITLE:

MATERIAL:

WEIGHT:

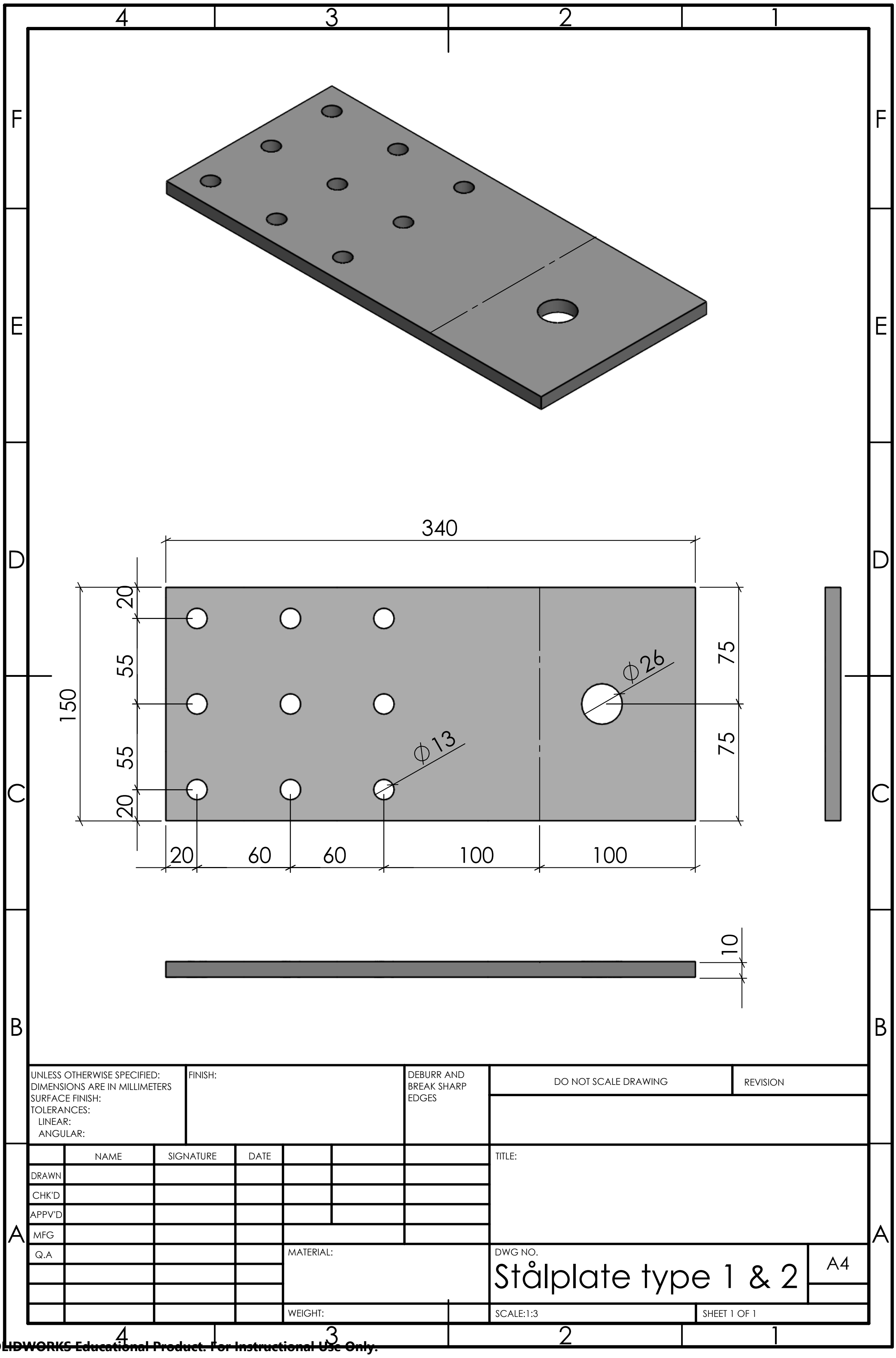
DWG NO.

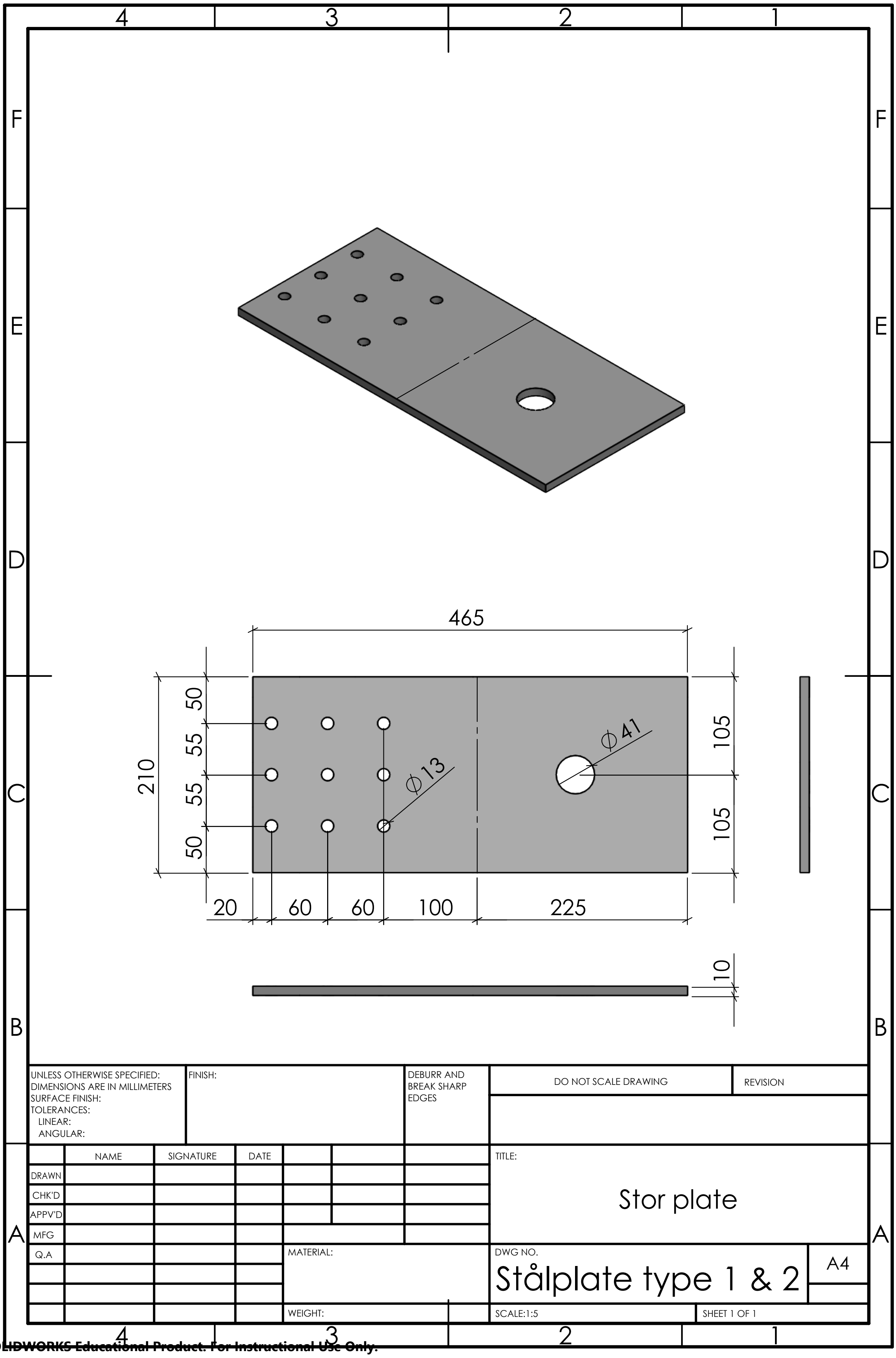
Specimen type 3

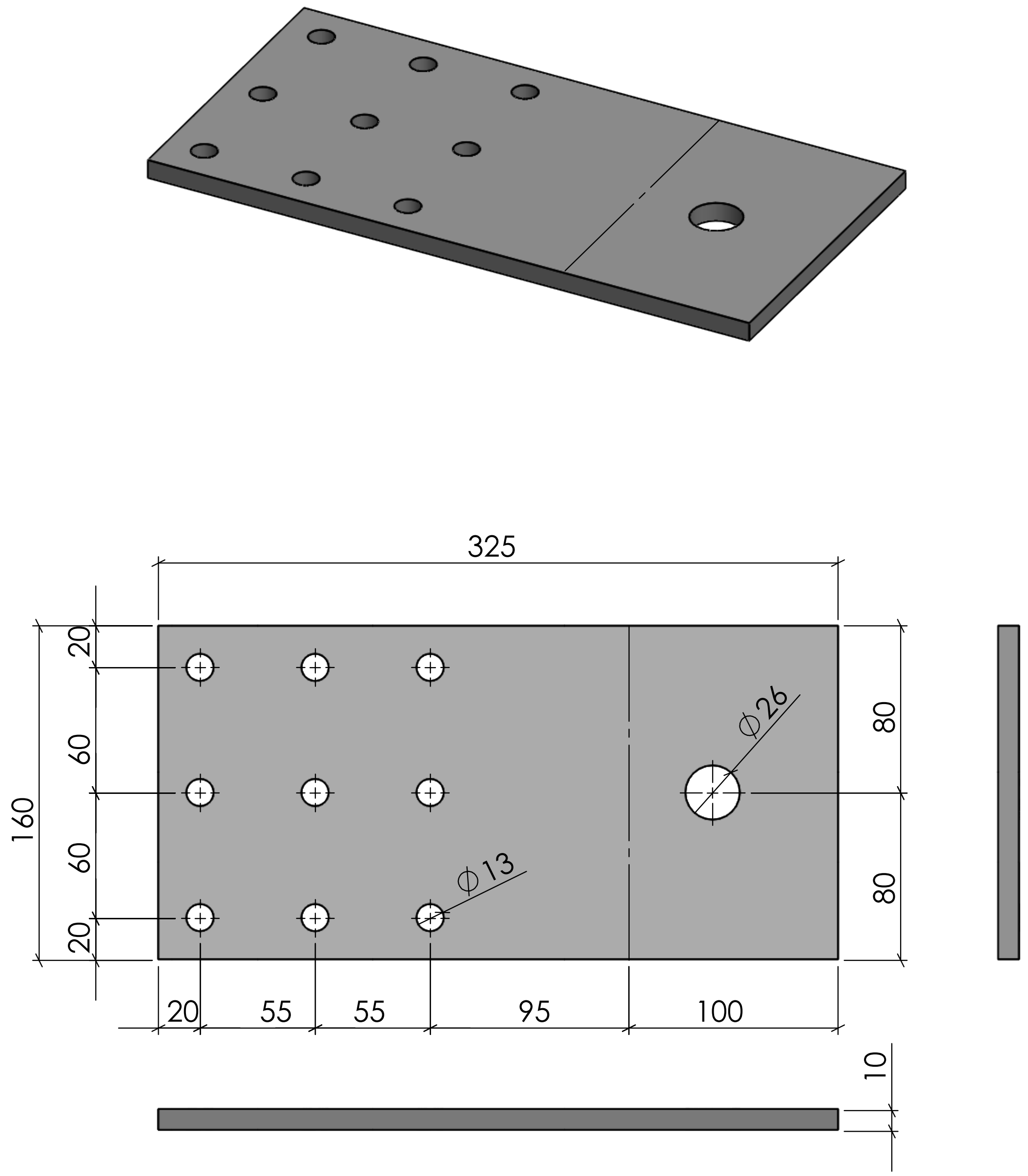
A4

SCALE:1:10

SHEET 1 OF 1


340
20
55
55
20
150
75
75
Φ26
Φ13
20
60
60
100
100
10
UNLESS OTHERWISE SPECIFIED:
DIMENSIONS ARE IN MILLIMETERS
SURFACE FINISH:
TOLERANCES:
LINEAR:
ANGULAR:
FINISH:
DEBURR AND BREAK SHARP EDGES
DO NOT SCALE DRAWING
REVISION
NAME
SIGNATURE
DATE
DRAWN
CHK'D
APPV'D
MFG
Q.A
TITLE:
MATERIAL:
WEIGHT:
DWG NO.
Stålplate type 1 & 2
A4
SCALE:1:3
SHEET 1 OF 1


465
210
50
55
55
50
20
60
60
100
225
105
105
Ø13
Ø41
10
UNLESS OTHERWISE SPECIFIED:
DIMENSIONS ARE IN MILLIMETERS
SURFACE FINISH:
TOLERANCES:
LINEAR:
ANGULAR:
FINISH:
DEBURR AND BREAK SHARP EDGES
DO NOT SCALE DRAWING
REVISION
NAME
SIGNATURE
DATE
DRAWN
CHK'D
APPV'D
MFG
Q.A
TITLE:
Stor plate
MATERIAL:
DWG NO.
Stålplate type 1 & 2
A4
WEIGHT:
SCALE:1:5
SHEET 1 OF 1



UNLESS OTHERWISE SPECIFIED:
DIMENSIONS ARE IN MILLIMETERS
SURFACE FINISH:
TOLERANCES:
LINEAR:
ANGULAR:

FINISH:

DEBURR AND BREAK SHARP EDGES

DO NOT SCALE DRAWING

REVISION

	NAME	SIGNATURE	DATE		
DRAWN					
CHK'D					
APPV'D					
MFG					
Q.A					

TITLE:

MATERIAL:

DWG NO.

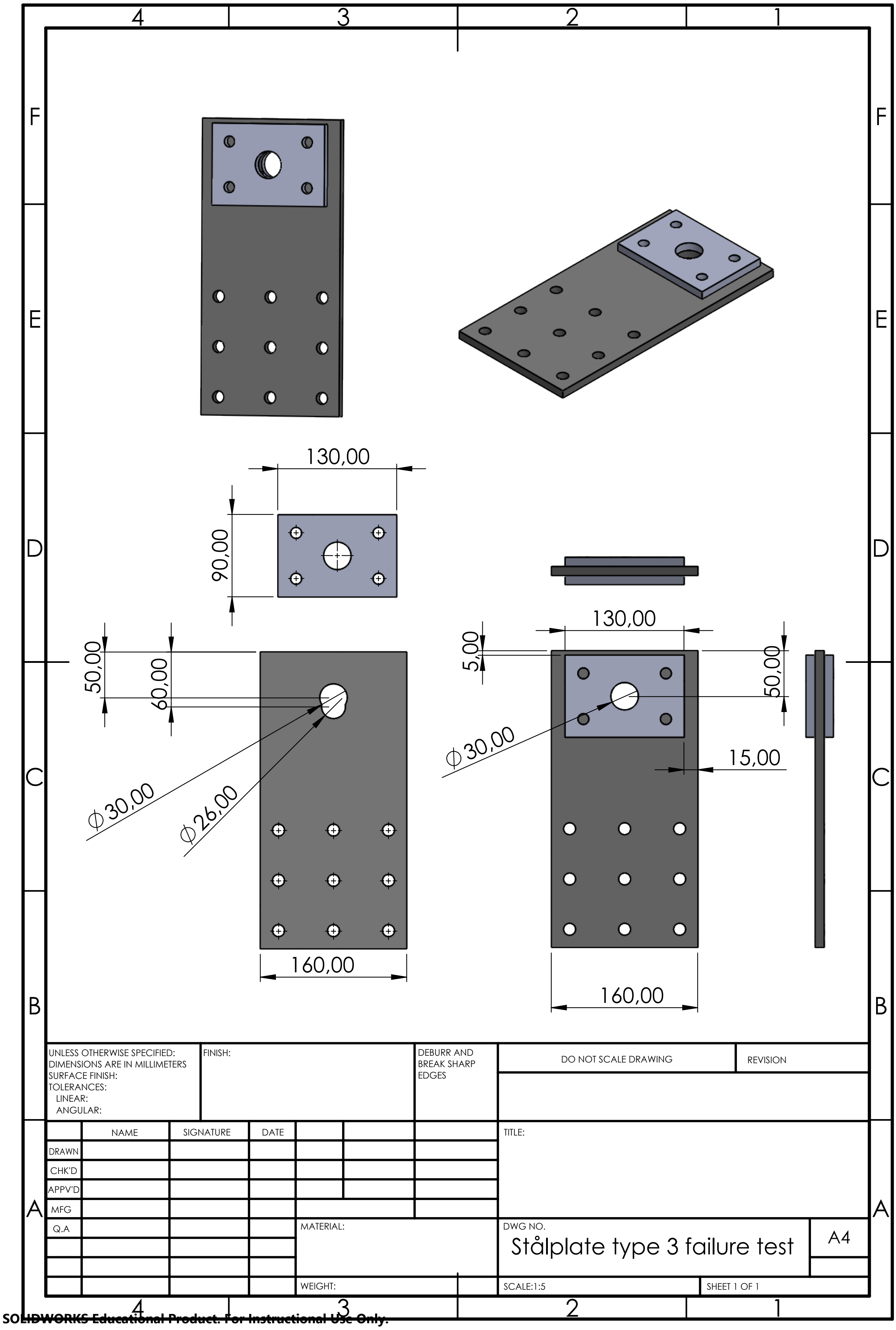
Stålplate type 3

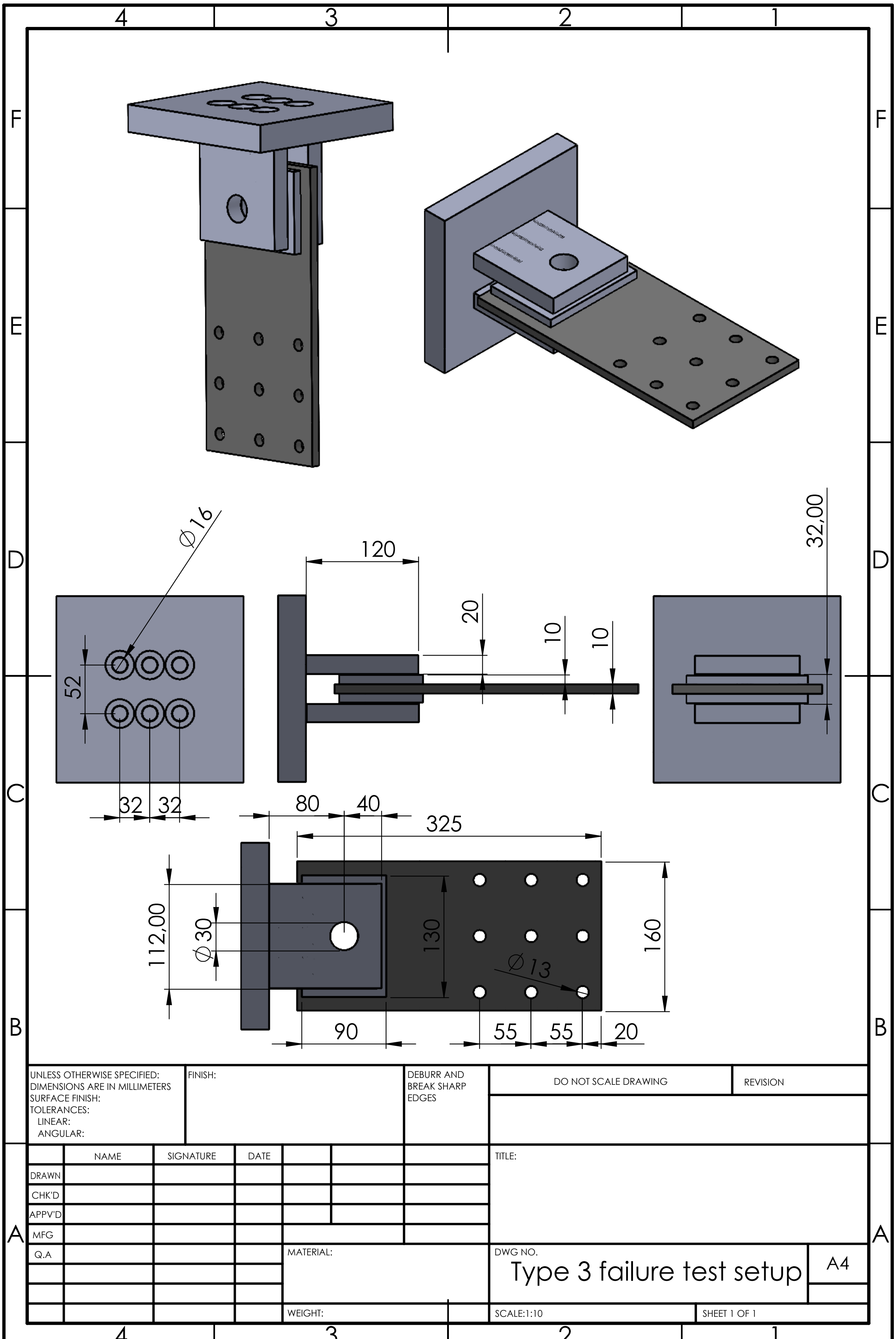
A4

WEIGHT:

SCALE:1:3

SHEET 1 OF 1


130,00
90,00
50,00
60,00
5,00
130,00
50,00
15,00
ϕ30,00
ϕ26,00
ϕ30,00
160,00
160,00
UNLESS OTHERWISE SPECIFIED:
DIMENSIONS ARE IN MILLIMETERS
SURFACE FINISH:
TOLERANCES:
LINEAR:
ANGULAR:
FINISH:
DEBURR AND BREAK SHARP EDGES
DO NOT SCALE DRAWING
REVISION
NAME
SIGNATURE
DATE
DRAWN
CHK'D
APPV'D
MFG
Q.A
TITLE:
MATERIAL:
DWG NO.
Stålplate type 3 failure test
A4
WEIGHT:
SCALE:1:5
SHEET 1 OF 1
SOLIDWORKS Educational Product. For Instructional Use Only.


4
3
2
1
F
E
D
C
B
A
Ø16
120
20
10
10
32,00
52
32
32
80
40
325
112,00
Ø30
130
160
Ø13
90
55
55
20
UNLESS OTHERWISE SPECIFIED:
DIMENSIONS ARE IN MILLIMETERS
SURFACE FINISH:
TOLERANCES:
LINEAR:
ANGULAR:
FINISH:
DEBURR AND
BREAK SHARP
EDGES
DO NOT SCALE DRAWING
REVISION
NAME
SIGNATURE
DATE
DRAWN
CHK'D
APPV'D
MFG
Q.A
TITLE:
MATERIAL:
DWG NO.
Type 3 failure test setup
A4
WEIGHT:
SCALE:1:10
SHEET 1 OF 1

F Calculations

This appendix includes all calculations that have been carried out throughout the project. All clauses used are referred to the Eurocode. An overview is given in table 6.

Table 6: Included documents in appendix F

Name	Description
Capacity check of steel plates Type 1 & 2 Original plates	Capacity calculation of the steel plates designed by Frette et. al (2021) for specimen type 1 & 2.
Capacity check of steel plates Type 1 & 2	Capacity calculation of the steel plate designed and used in the experimental work for specimen type 1 and 2. The plate was designed in order to recist a design force of 400 kN.
Capacity check of steel plates Type 3	Capacity calculation of steel plate designed by Frette et. al (2021) for specimen type 3. Checked for a design force of 100 kN.
Capacity check of steel plates Type 3 Failure test	Capacity calculation of steel plate modified in order to be used in failure test of specimen type 3 in the 250 kN actuator.
Capacity check of timber parts	Capacity calculation of all timber specimens to estimate their capacities to be used in load procedure. The calculation document also includes the theoretical stiffness calculations.
Capacity check of anchoring plates	Capacity calculation of the designed anchoring plates that were used in the 400 kN actuator. Of that reason, the plates were designed with 400 kN as design force.
Loading procedure: Elastic domain Specimen type 1	Calculation of all load levels, mean load levels and load amplitudes applied to specimen type 1. Estimated load capacity was based on calculation note Capacity check of timber parts.
Loading procedure: Elastic domain Specimen type 2	Calculation of all load levels, mean load levels and load amplitudes applied to specimen type 2. Estimated load capacity was based on calculation note Capacity check of timber parts.
Loading procedure: Elastic domain Specimen type 3	Calculation of all load levels, mean load levels and load amplitudes applied to specimen type 3. Estimated load capacity was based on calculation note Capacity check of timber parts.

Capacity check of steel plates | Type 1 & 2 Original plates

Minimum distances:
EC3-1-8, Table 3.3

Calculating both optimal and minimum distances.

$d := 26\ mm$ (Outer bolt diameter)

$d_0 := d = 26\ mm$ (Bore holde diameter in steel plate)

Optimal distances:

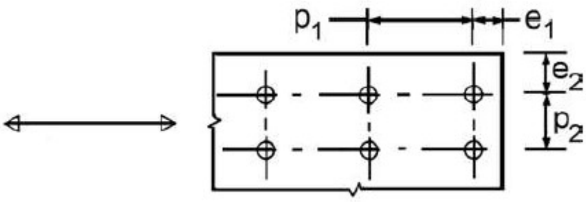
$e_{1.optimal} := 3 \cdot d_0 = 78\ mm$

$p_{1.optimal} := 3.75 \cdot d_0 = 97.5\ mm$

$e_{2.optimal} := 1.5 \cdot d_0 = 39\ mm$

$p_{2.optimal} := 3 \cdot d_0 = 78\ mm$

Minimum distances:



$e_{1.min} := 1.2 \cdot d_0 = 31.2\ mm$

$p_{1.min} := 2.2 \cdot d_0 = 57.2\ mm$

$e_{2.min} := 1.2 \cdot d_0 = 31.2\ mm$

$p_{2.min} := 2.4 \cdot d_0 = 62.4\ mm$

Material properties

Due to geometrical restrictions on the hydraulick jack, the optimal distances may not be chosen. Controlling the capacity for the following chosen, relevant values.

$e_1 := 60\ mm$ (End distance in force direction)

$e_2 := 75\ mm$ (End distance perpendicular to force direction)

$f_{ub} := 470\ \frac{N}{mm^2}$ (Ultimate stress of bolt)

$f_u := 470\ \frac{N}{mm^2}$ (Ulitmate stress of steel plate)

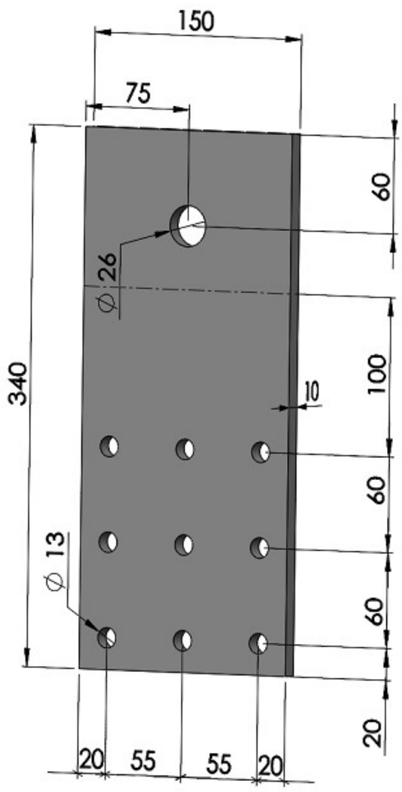
$t := 10\ mm$ (Thickness of steel plates)

$b := 2 \cdot e_2 = 150\ mm$ (Width of steel plate)

$f_y := 355\ \frac{N}{mm^2}$ (Yielding stress of steel plate)

$\gamma_{M2} := 1.25$ (Material factor for steel in connections)

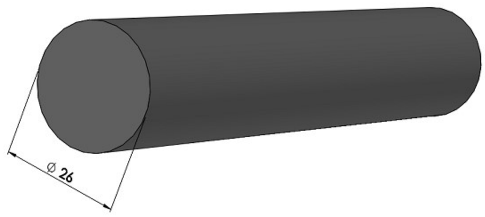
$\gamma_{M0} := 1.05$ (Material factor for steel)



Shear capacity of bolt:
EC3-1-8, Table 3.4

$\alpha_v := 0.6$

$d = 26\ mm$



$A := \frac{\pi}{4} \cdot d^2 = 530.929\ mm^2$

$n_{shear\_planes} := 2$

$F_{v.Rd} := n_{shear\_planes} \cdot \frac{\alpha_v \cdot f_{ub} \cdot A}{\gamma_{M2}} = 239.555\ kN$

Bearing capacity:
EC3-1-8, Table 3.4

$t_{extra\_steel} := 8\ mm$ (Extra steel welded-on on each side of steel plate in order to avoid eccentricity in fork and increase bearing capacity.)

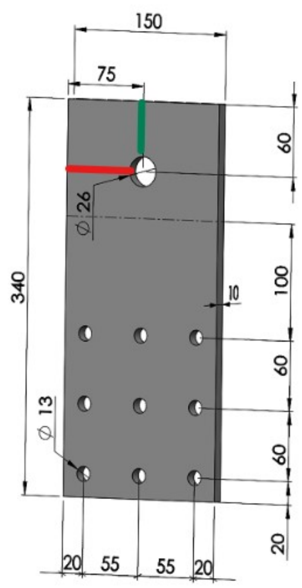
$t_{tot} := t + 2 \cdot t_{extra\_steel} = 26\ mm$

$\alpha_b := min\left(\frac{e_1}{3 \cdot d_0}, \frac{f_{ub}}{f_u}, 1.0\right) = 0.769$

$k_1 := min\left(2.8 \cdot \frac{e_2}{d_0} - 1.7, 2.5\right) = 2.5$

$F_{b.Rd} := \frac{k_1 \cdot \alpha_b \cdot f_u \cdot d \cdot t_{tot}}{\gamma_{M2}} = 488.8\ kN$

Block tearing:
EC3-1-8, 3.10.2



$A_{nt} := \left(e_2 - \frac{d_0}{2}\right) \cdot t = 620\ mm^2$

$A_{nv} := \left(e_1 - \frac{d_0}{2}\right) \cdot t = 470\ mm^2$

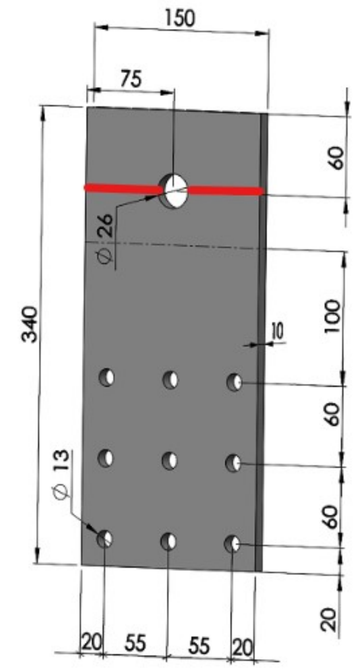
$V_{eff.1.Rd} := \frac{f_u \cdot A_{nt}}{\gamma_{M2}} + \left(\frac{1}{\sqrt{3}}\right) \cdot \frac{f_y \cdot A_{nv}}{\gamma_{M0}} = 324.864\ kN$

Tension:
EC3-1-1, 3.10.2

$A := b \cdot t = \left(1.5 \cdot 10^3\right)\ mm^2$

$A_{net} := A - d_0 \cdot t = \left(1.24 \cdot 10^3\right)\ mm^2$

$N_{t.Rd} := min\left(\frac{A \cdot f_y}{\gamma_{M0}}, \frac{0.9 \cdot A_{net} \cdot f_u}{\gamma_{M2}}\right) = 419.616\ kN$



Capacity check of steel plates | Type 1 & 2

Minimum distances:
EC3-1-8, Table 3.3

Calculating both optimal and minimum distances.

$d := 40\ mm$ (Outer bolt diameter)

$d_0 := d + mm = 41\ mm$ (Bore holde diameter in steel plate)

Optimal distances:

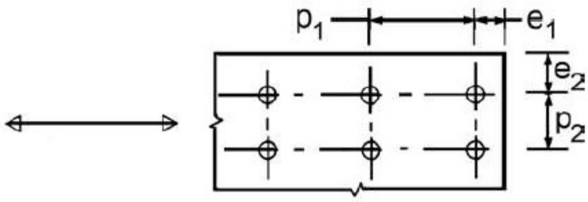
$e_{1.optimal} := 3 \cdot d_0 = 123\ mm$

$p_{1.optimal} := 3.75 \cdot d_0 = 153.75\ mm$

$e_{2.optimal} := 1.5 \cdot d_0 = 61.5\ mm$

$p_{2.optimal} := 3 \cdot d_0 = 123\ mm$

Minimum distances:



$e_{1.min} := 1.2 \cdot d_0 = 49.2\ mm$

$p_{1.min} := 2.2 \cdot d_0 = 90.2\ mm$

$e_{2.min} := 1.2 \cdot d_0 = 49.2\ mm$

$p_{2.min} := 2.4 \cdot d_0 = 98.4\ mm$

Material properties

Due to geometrical restrictions on the hydraulick jack, the optimal distances may not be chosen. Controlling the capacity for the following chosen, relevant values.

$e_1 := 120\ mm$ (End distance in force direction)

$e_2 := 105\ mm$ (End distance perpendicular to force direction)

$f_{ub} := 470\ \frac{N}{mm^2}$ (Ultimate stress of bolt)

$f_u := 470\ \frac{N}{mm^2}$ (Ulitmate stress of steel plate)

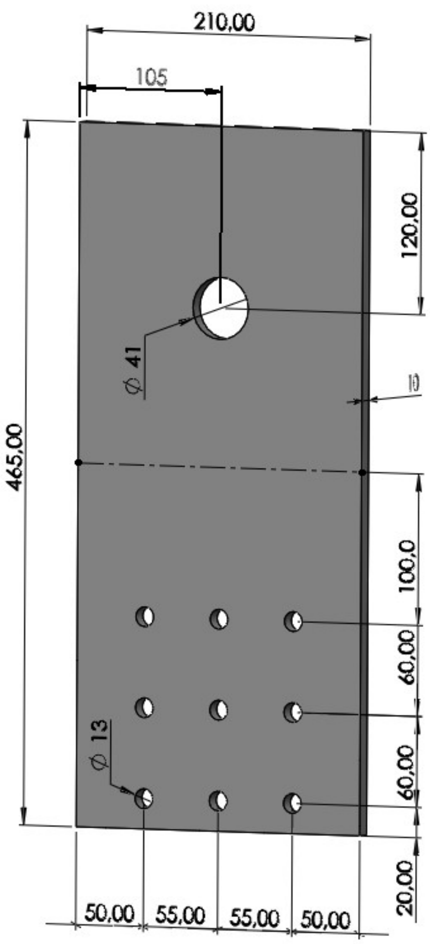
$t := 10\ mm$ (Thickness of steel plates)

$b := 2 \cdot e_2 = 210\ mm$ (Width of steel plate)

$f_y := 355\ \frac{N}{mm^2}$ (Yielding stress of steel plate)

$\gamma_{M2} := 1.25$ (Material factor for steel in connections)

$\gamma_{M0} := 1.05$ (Material factor for steel)



Shear capacity of bolt:
EC3-1-8, Table 3.4

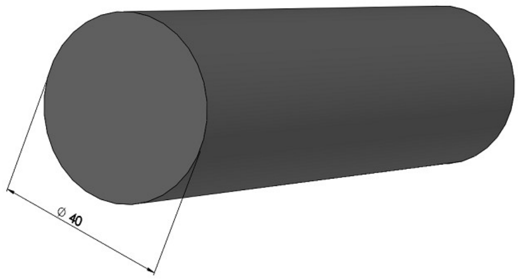
$\alpha_v := 0.6$

$d = 40\ mm$

$A := \frac{\pi}{4} \cdot d^2 = \left(1.257 \cdot 10^3\right)\ mm^2$

$n_{shear\_planes} := 2$

$F_{v.Rd} := n_{shear\_planes} \cdot \frac{\alpha_v \cdot f_{ub} \cdot A}{\gamma_{M2}} = 566.995\ kN$



Bearing capacity:
EC3-1-8, Table 3.4

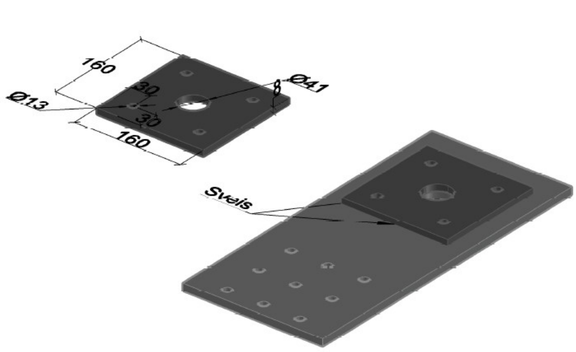
$t_{extra\_steel} := 8\ mm$ (Extra steel welded-on on each side of steel plate in order to avoid eccentricity in fork and increase bearing capacity.)

$t_{tot} := t + 2 \cdot t_{extra\_steel} = 26\ mm$

$\alpha_b := min\left(\frac{e_1}{3 \cdot d_0}, \frac{f_{ub}}{f_u}, 1.0\right) = 0.976$

$k_1 := min\left(2.8 \cdot \frac{e_2}{d_0} - 1.7, 2.5\right) = 2.5$

$F_{b.Rd} := \frac{k_1 \cdot \alpha_b \cdot f_u \cdot d \cdot t_{tot}}{\gamma_{M2}} = 953.756\ kN$

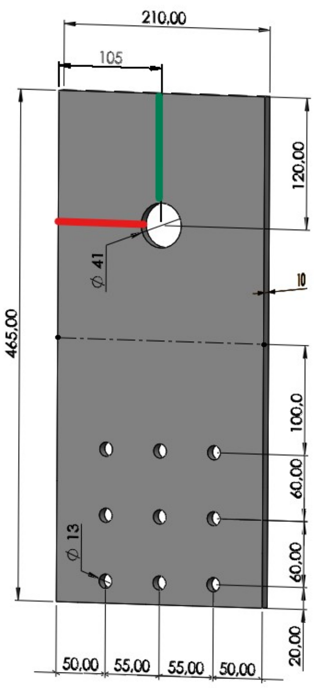


Block tearing:
EC3-1-8, 3.10.2

$A_{nt} := \left(e_2 - \frac{d_0}{2}\right) \cdot t = 845\ mm^2$

$A_{nv} := \left(e_1 - \frac{d_0}{2}\right) \cdot t = 995\ mm^2$

$V_{eff.1.Rd} := \frac{f_u \cdot A_{nt}}{\gamma_{M2}} + \left(\frac{1}{\sqrt{3}}\right) \cdot \frac{f_y \cdot A_{nv}}{\gamma_{M0}} = 511.943\ kN$

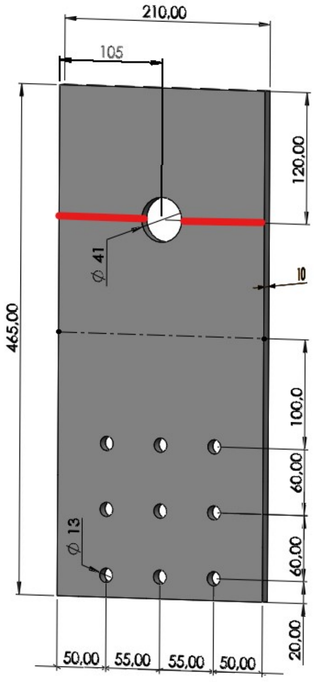


Tension:
EC3-1-1, 3.10.2

$A := b \cdot t = \left(2.1 \cdot 10^3\right)\ mm^2$

$A_{net} := A - d_0 \cdot t = \left(1.69 \cdot 10^3\right)\ mm^2$

$N_{t.Rd} := min\left(\dfrac{A \cdot f_y}{\gamma_{M0}}, \dfrac{0.9 \cdot A_{net} \cdot f_u}{\gamma_{M2}}\right) = 571.896\ kN$



Capacity check of steel plates | Type 3

Minimum distances:
EC3-1-8, Table 3.3

Calculating both optimal and minimum distances.

$d := 26\ mm$ (Outer pipe diameter)

$d_0 := d = 26\ mm$ (Bore holde diameter in steel plate)

Optimal distances:

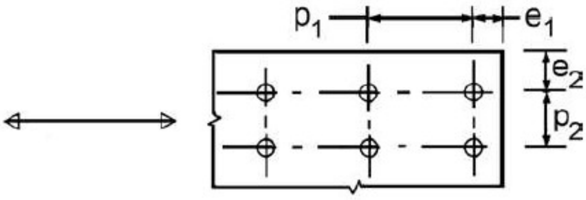
$e_{1.optimal} := 3 \cdot d_0 = 78\ mm$

$p_{1.optimal} := 3.75 \cdot d_0 = 97.5\ mm$

$e_{2.optimal} := 1.5 \cdot d_0 = 39\ mm$

$p_{2.optimal} := 3 \cdot d_0 = 78\ mm$

Minimum distances:



$e_{1.min} := 1.2 \cdot d_0 = 31.2\ mm$

$p_{1.min} := 2.2 \cdot d_0 = 57.2\ mm$

$e_{2.min} := 1.2 \cdot d_0 = 31.2\ mm$

$p_{2.min} := 2.4 \cdot d_0 = 62.4\ mm$

Material properties

Due to geometrical restrictions on the hydraulick jack, the optimal distances may not be chosen. Controlling the capacity for the following chosen, relevant values.

$e_1 := 60\ mm$ (End distance in force direction)

$e_2 := 80\ mm$ (End distance perpendicular to force direction)

$f_{ub} := 470\ \frac{N}{mm^2}$ (Ultimate stress of bolt)

$f_u := 470\ \frac{N}{mm^2}$ (Ulitmate stress of steel plate)

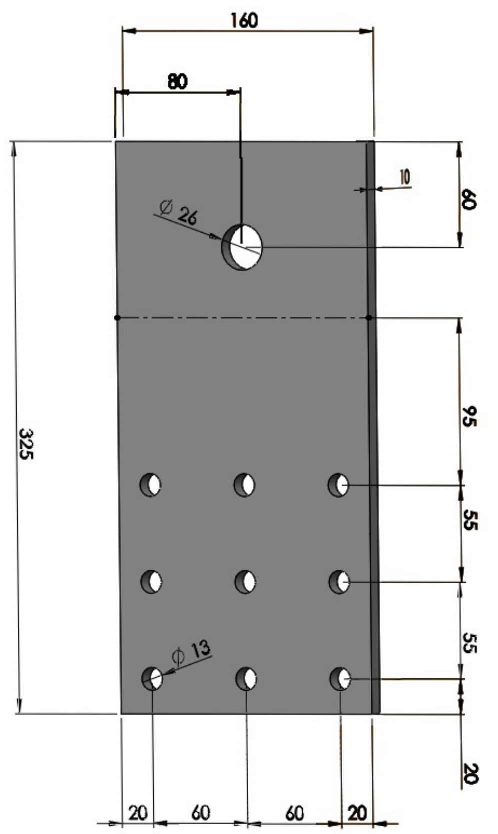
$t := 10\ mm$ (Thickness of steel plates)

$b := 2 \cdot e_2 = 160\ mm$ (Width of steel plate)

$f_y := 355\ \frac{N}{mm^2}$ (Yielding stress of steel plate)

$\gamma_{M2} := 1.25$ (Material factor for steel in connections)

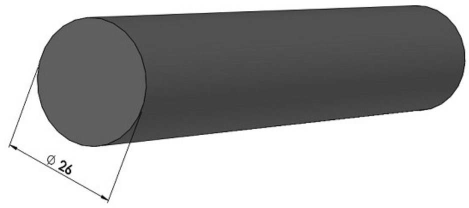
$\gamma_{M0} := 1.05$ (Material factor for steel)



Shear capacity of bolt:
EC3-1-8, Table 3.4

$\alpha_v := 0.6$

$d = 26\ mm$



$A := \frac{\pi}{4} \cdot d^2 = 530.929\ mm^2$

$n_{shear\_planes} := 2$

$F_{v.Rd} := n_{shear\_planes} \cdot \frac{\alpha_v \cdot f_{ub} \cdot A}{\gamma_{M2}} = 239.555\ kN$

Bearing capacity:
EC3-1-8, Table 3.4

$\alpha_b := min\left(\frac{e_1}{3 \cdot d_0}, \frac{f_{ub}}{f_u}, 1.0\right) = 0.769$

$k_1 := min\left(2.8 \cdot \frac{e_2}{d_0} - 1.7, 2.5\right) = 2.5$

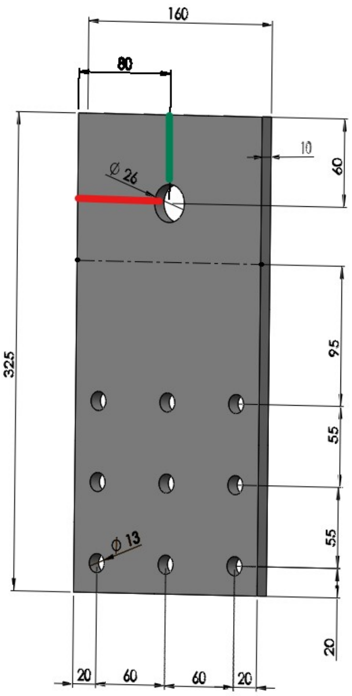
$F_{b.Rd} := \frac{k_1 \cdot \alpha_b \cdot f_u \cdot d \cdot t}{\gamma_{M2}} = 188\ kN$

Block tearing:
EC3-1-8, 3.10.2

$A_{nt} := \left(e_2 - \frac{d_0}{2}\right) \cdot t = 670\ mm^2$

$A_{nv} := \left(e_1 - \frac{d_0}{2}\right) \cdot t = 470\ mm^2$

$V_{eff.1.Rd} := \frac{f_u \cdot A_{nt}}{\gamma_{M2}} + \left(\frac{1}{\sqrt{3}}\right) \cdot \frac{f_y \cdot A_{nv}}{\gamma_{M0}} = 343.664\ kN$



Tension:
EC3-1-1, 3.10.2

$A := b \cdot t = \left(1.6 \cdot 10^{3}\right)\ mm^{2}$

$A_{net} := A - d_0 \cdot t = \left(1.34 \cdot 10^{3}\right)\ mm^{2}$

$$N_{t.Rd} := min\left(\frac{A \cdot f_y}{\gamma_{M0}}, \frac{0.9 \cdot A_{net} \cdot f_u}{\gamma_{M2}}\right) = 453.456\ kN$$

Capacity check of steel plates | Type 3 Failure test

Minimum distances:
EC3-1-8, Table 3.3

Calculating both optimal and minimum distances.

$d := 30\ mm$ (Outer pipe diameter)

$d_0 := d = 30\ mm$ (Bore holde diameter in steel plate)

Optimal distances:

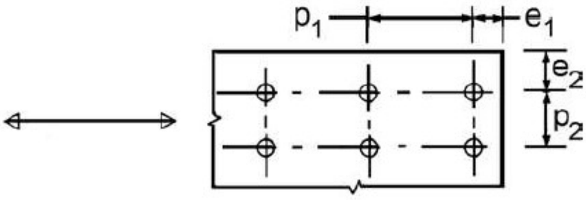
$e_{1.optimal} := 3 \cdot d_0 = 90\ mm$

$p_{1.optimal} := 3.75 \cdot d_0 = 112.5\ mm$

$e_{2.optimal} := 1.5 \cdot d_0 = 45\ mm$

$p_{2.optimal} := 3 \cdot d_0 = 90\ mm$

Minimum distances:



$e_{1.min} := 1.2 \cdot d_0 = 36\ mm$

$p_{1.min} := 2.2 \cdot d_0 = 66\ mm$

$e_{2.min} := 1.2 \cdot d_0 = 36\ mm$

$p_{2.min} := 2.4 \cdot d_0 = 72\ mm$

Material properties

Due to geometrical restrictions on the hydraulick jack, the optimal distances may not be chosen. Controlling the capacity for the following chosen, relevant values.

$e_1 := 50\ mm$ (End distance in force direction)

$e_2 := 80\ mm$ (End distance perpendicular to force direction)

$f_{ub} := 470\ \frac{N}{mm^2}$ (Ultimate stress of bolt)

$f_u := 470\ \frac{N}{mm^2}$ (Ulitmate stress of steel plate)

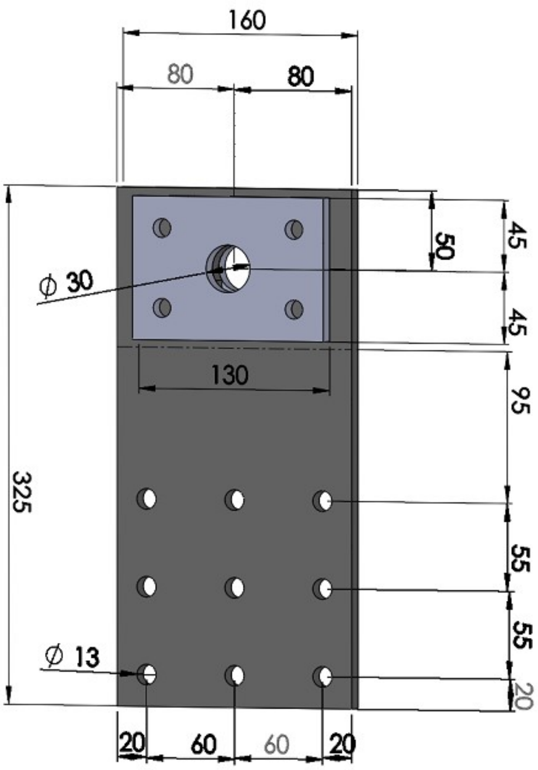
$t := 10\ mm$ (Thickness of steel plates)

$b := 2 \cdot e_2 = 160\ mm$ (Width of steel plate)

$f_y := 355\ \frac{N}{mm^2}$ (Yielding stress of steel plate)

$\gamma_{M2} := 1.25$ (Material factor for steel in connections)

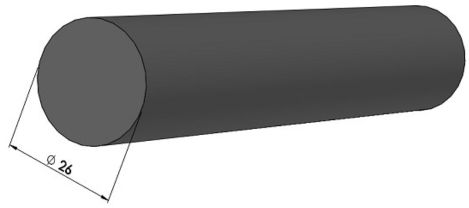
$\gamma_{M0} := 1.05$ (Material factor for steel)



Shear capacity of bolt:
EC3-1-8, Table 3.4

$\alpha_v := 0.6$

$d = 30\ mm$



$A := \frac{\pi}{4} \cdot d^2 = 706.858\ mm^2$

$n_{shear\_planes} := 2$

$F_{v.Rd} := n_{shear\_planes} \cdot \frac{\alpha_v \cdot f_{ub} \cdot A}{\gamma_{M2}} = 318.934\ kN$

Bearing capacity:
EC3-1-8, Table 3.4

$\alpha_b := min\left(\frac{e_1}{3 \cdot d_0}, \frac{f_{ub}}{f_u}, 1.0\right) = 0.556$

$k_1 := min\left(2.8 \cdot \frac{e_2}{d_0} - 1.7, 2.5\right) = 2.5$

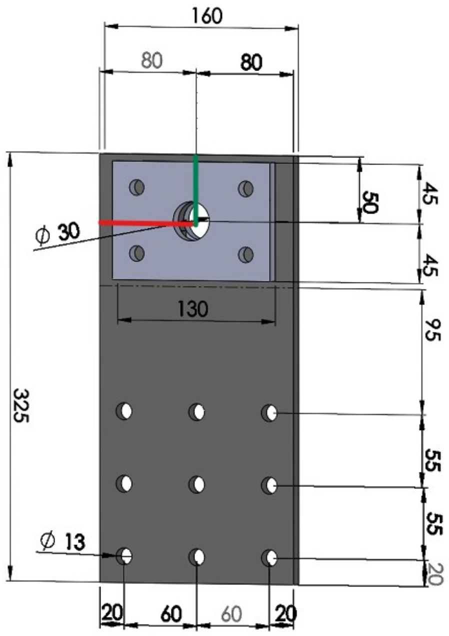
$F_{b.Rd} := \frac{k_1 \cdot \alpha_b \cdot f_u \cdot d \cdot t}{\gamma_{M2}} = 156.667\ kN$

Block tearing:
EC3-1-8, 3.10.2

$A_{nt} := \left(e_2 - \frac{d_0}{2}\right) \cdot t = 650\ mm^2$

$A_{nv} := \left(e_1 - \frac{d_0}{2}\right) \cdot t = 350\ mm^2$

$V_{eff.1.Rd} := \frac{f_u \cdot A_{nt}}{\gamma_{M2}} + \left(\frac{1}{\sqrt{3}}\right) \cdot \frac{f_y \cdot A_{nv}}{\gamma_{M0}} = 312.72\ kN$



Tension:
EC3-1-1, 3.10.2

$$A := b \cdot t = \left(1.6 \cdot 10^{3}\right)\ mm^{2}$$

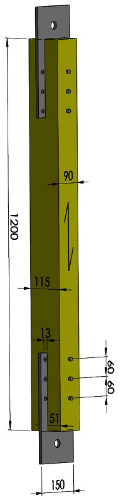
$$A_{net} := A - d_0 \cdot t = \left(1.3 \cdot 10^{3}\right)\ mm^{2}$$

$$N_{t.Rd} := min\left(\frac{A \cdot f_y}{\gamma_{M0}}, \frac{0.9 \cdot A_{net} \cdot f_u}{\gamma_{M2}}\right) = 439.92\ kN$$

Capacity check of timber parts

Splitting ǁ grain
EC5-1-1, 8.5.1

Connection: Type 1 (T22)



$d := 12\ mm$ (Dowel diameter)

$f_{uk} := 916\ MPa$ (Ultimate stress dowel)

$M_{y.Rk} := 0.30 \cdot f_{uk} \cdot \left(\frac{d}{mm}\right)^{2.6} \cdot mm^3 = 0.176\ kN \cdot m$ (Yielding moment of dowel)

$\alpha := 0\ °$ (Angle to grain)

$k_{90} := 1.35 + 0.015 \cdot \left(\frac{d}{mm}\right) = 1.53$ (Modification factor)

$\rho_{mean} := 470\ \frac{kg}{m^3}$ (Characteristic timber density)

$f_{h.0.k} := 0.082 \cdot \left(1 - 0.01 \cdot \left(\frac{d}{mm}\right)\right) \cdot \left(\frac{\rho_{mean}}{\frac{kg}{m^3}}\right) \cdot MPa = 33.915\ MPa$ (Timber strength)

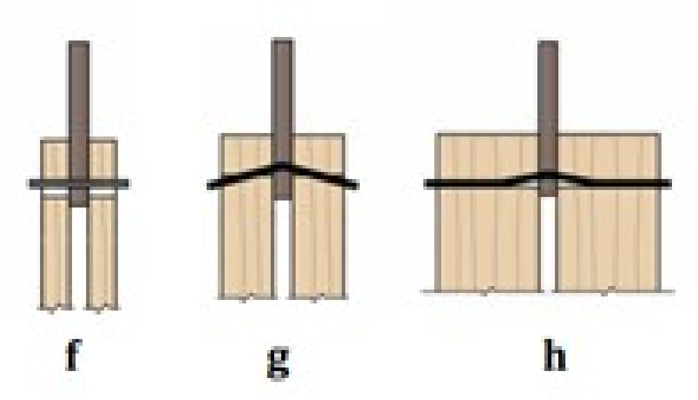
$f_{h.\alpha.k} := \frac{f_{h.0.k}}{k_{90} \cdot \sin(\alpha)^2 + \cos(\alpha)^2} = 33.915\ MPa$ (Angle-to-grain timber strength)

$a_1 := 60\ mm$ (Spacing between dowels in fibre direction)

$n := 3$ (Number of fasteners in a row)

$n_{ef} := min\left(n, n^{0.9} \cdot \sqrt[4]{\frac{a_1}{13 \cdot d}}\right) = 2.117$ (Effective number of fasteners)

$t_1 := 51\ mm$ (Thickness of side member)



$F_{v.Rk.f} := f_{h.\alpha.k} \cdot t_1 \cdot d = 20.756\ kN$ (Failure mode (f))

$F_{v.Rk.g} := f_{h.\alpha.k} \cdot t_1 \cdot d \cdot \left(\sqrt{2 + \frac{4 \cdot M_{y.Rk}}{f_{h.\alpha.k} \cdot d \cdot {t_1}^2}} - 1 \right) = 13.122\ kN$ (Failure mode (g))

$F_{v.Rk.h} := 2.3 \cdot \sqrt{M_{y.Rk} \cdot f_{h.\alpha.k} \cdot d} = 19.452\ kN$ (Failure mode (h))

$n_{columns} := 1$ (Number of columns/number of rows of fasteneres)

$n_{plates} := 1$ (Number of internal steel plates)

$F_{v.Rk.brittle} := n_{ef} \cdot n_{columns} \cdot n_{plates} \cdot F_{v.Rk.f} = 43.935\ kN$ (Brittle capacity)

$F_{v.Rk.ductile} := n_{ef} \cdot n_{columns} \cdot n_{plates} \cdot min\left(F_{v.Rk.g}, F_{v.Rk.h}\right) = 27.776\ kN$ (Ductile capacity)

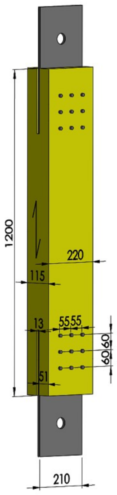
$n_{connections} := 2$ (Number of connections)

Final connection capacity:

$F_{v.Rk} := n_{connections} \cdot min\left(F_{v.Rk.brittle}, F_{v.Rk.ductile}\right) = 55.552\ kN$ (Final capacity)

Splitting ∥ grain
EC5-1-1, 8.5.1

Connection: Type 2 (T15)



$d := 12\ mm$ (Dowel diameter)

$f_{uk} := 916\ MPa$ (Ultimate stress dowel)

$M_{y.Rk} := 0.30 \cdot f_{uk} \cdot \left(\frac{d}{mm}\right)^{2.6} \cdot mm^3 = 0.176\ kN \cdot m$ (Yielding moment of dowel)

$\alpha := 0\ °$ (Angle to grain)

$k_{90} := 1.35 + 0.015 \cdot \left(\frac{d}{mm}\right) = 1.53$ (Modification factor)

$\rho_{mean} := 430\ \frac{kg}{m^3}$ (Mean timber density)

$f_{h.0.k} := 0.082 \cdot \left(1 - 0.01 \cdot \left(\frac{d}{mm}\right)\right) \cdot \left(\frac{\rho_{mean}}{\frac{kg}{m^3}}\right) \cdot MPa = 31.029\ MPa$ (Timber strength)

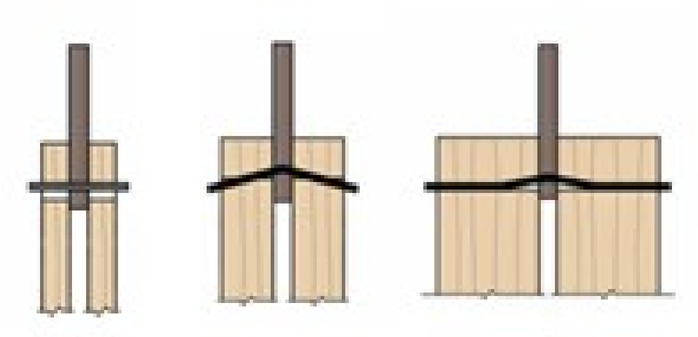
$f_{h.\alpha.k} := \frac{f_{h.0.k}}{k_{90} \cdot \sin(\alpha)^2 + \cos(\alpha)^2} = 31.029\ MPa$ (Angle-to-grain timber strength)

$a_1 := 60\ mm$ (Spacing between dowels in fibre direction)

$n := 3$ (Number of fasteners in a row)

$n_{ef} := min\left(n, n^{0.9} \cdot \sqrt[4]{\frac{a_1}{13 \cdot d}}\right) = 2.117$ (Effective number of fasteners)

$t_1 := 51\ mm$ (Thickness of side member)



$F_{v.Rk.f} := f_{h.\alpha.k} \cdot t_1 \cdot d = 18.99\ kN$ (Failure mode (f))

$F_{v.Rk.g} := f_{h.\alpha.k} \cdot t_1 \cdot d \cdot \left(\sqrt{2 + \frac{4 \cdot M_{y.Rk}}{f_{h.\alpha.k} \cdot d \cdot {t_1}^2}} - 1 \right) = 12.363\ kN$ (Failure mode (g))

$F_{v.Rk.h} := 2.3 \cdot \sqrt{M_{y.Rk} \cdot f_{h.\alpha.k} \cdot d} = 18.606\ kN$ (Failure mode (h))

$n_{columns} := 3$ (Number of columns/number of rows of fasteneres)

$n_{plates} := 1$ (Number of internal steel plates)

$F_{v.Rk.brittle} := n_{ef} \cdot n_{columns} \cdot n_{plates} \cdot F_{v.Rk.f} = 120.588\ kN$ (Brittle capacity)

$F_{v.Rk.ductile} := n_{ef} \cdot n_{columns} \cdot n_{plates} \cdot min\left(F_{v.Rk.g}, F_{v.Rk.h}\right) = 78.505\ kN$ (Ductile capacity)

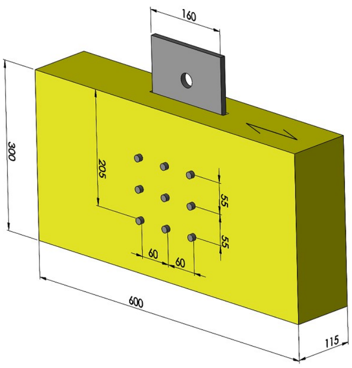
$n_{connections} := 2$ (Number of connections)

Final connection capacity:

$F_{v.Rk} := n_{connections} \cdot min\left(F_{v.Rk.brittle}, F_{v.Rk.ductile}\right) = 157.011\ kN$ (Final capacity)

Splitting perpendicular to grain
EC5-1-1, 8.5.1

Connection: Type 3 (T15)



$h_e := 205\ mm$ (Most distanced fastener)

$h := 300\ mm$ (Height of specimen)

$b := 102\ mm$ (Width of specimen)

$w := 1.0$ (Modification factor)

$$F_{90.Rk} := 14 \cdot \frac{b}{m} \cdot w \cdot \sqrt{\frac{\frac{h_e}{mm}}{\left(1 - \frac{h_e}{h}\right)}} \cdot kN = 36.333\ kN$$ (Capacity)

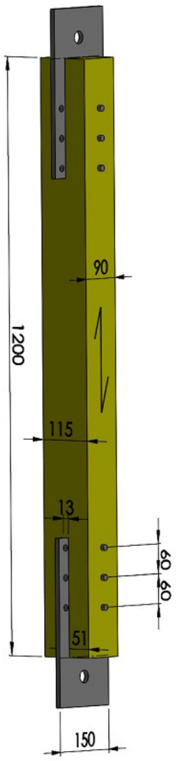
$n_{shearplanes} := 2$ (Number of shear planes)

Final connection capacity:

$F_{v.90.Rk} := n_{shearplanes} \cdot F_{90.Rk} = 72.666\ kN$ (Final capacity)

Tension ∥ grain
EC5-1-1, 6.1.2

Connection: Type 1 (T22)



$d = 12\ mm$ (Dowel diameter)

$b := 115\ mm$ (Width of wood)

$h := 90\ mm$ (Heigth of wood)

$t_{slot} := 13\ mm$ (Thickness of slot)

$n_{columns} := 1$ (Number of columns of fasteners)

$t_1 = 51\ mm$ (Thickness of side member)

$A_{net} := b \cdot h - 2 \cdot n_{columns} \cdot d \cdot t_1 - t_{slot} \cdot h = \left(7.956 \cdot 10^3\right)\ mm^2$ (Net area)

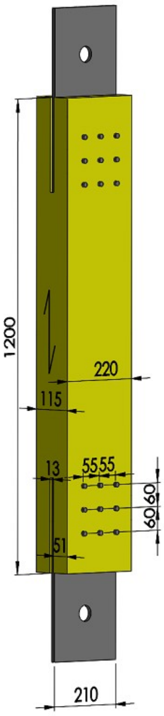
$f_{t.0.k} := 15\ MPa$ (Timber strength in fibre direction)

Final connection capacity:

$F_{Rk} := A_{net} \cdot f_{t.0.k} = 119.34\ kN$ (Final capacity)

Tension || grain
EC5-1-1, 6.1.2

Connection: Type 2 (T15)



$d=12\ mm$ (Dowel diameter)

$b:=115\ mm$ (Width of wood)

$h:=220\ mm$ (Heigth of wood)

$t_{slot}:=13\ mm$ (Thickness of slot)

$n_{columns}:=3$ (Number of columns of fasteners)

$t_1=51\ mm$ (Thickness of side member)

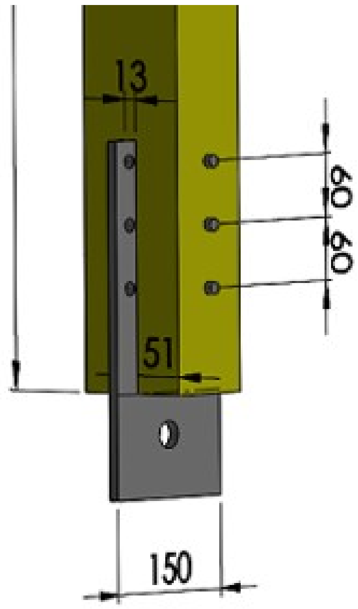
$A_{net}:=b\cdot h-2\cdot n_{columns}\cdot d\cdot t_1-t_{slot}\cdot h=\left(1.877\cdot 10^4\right)\ mm^2$ (Net area)

$f_{t.0.k}:=15\ MPa$ (Timber strength in fibre direction)

Final connection capacity:

$F_{Rk}:=A_{net}\cdot f_{t.0.k}=281.52\ kN$ (Final capacity)

Connection stiffness
EC5-1-1, 7.1
Connection: Type 1 (T22)



$\rho_{mean} := 470 \, \frac{kg}{m^3}$ (Mean density)

$d = 12 \, mm$ (Dowel diameter)

$K_{ser} := 2 \cdot \frac{\rho_{mean}^{(1.5)} \cdot d}{23} \cdot \left(\frac{1}{\left(\frac{kg}{m^3} \right)^{(1.5)}} \right) \cdot \left(\frac{1}{m} \right) \frac{kN}{mm} = 10.632 \, \frac{kN}{mm}$ (Stiffness per fastener per shear plane)

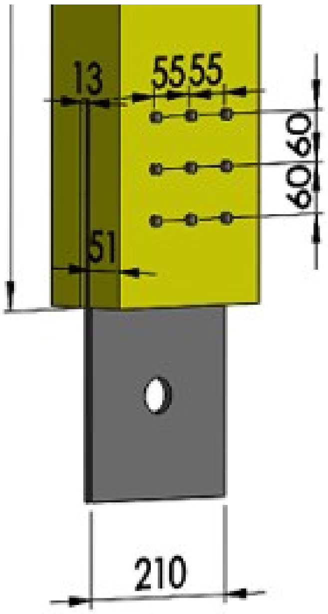
Final connection stiffness:

$n_{fasteners} := 3$ (Number of fasteners)

$n_{shearplanes} := 2$ (Number of shear planes)

$K_{ser.final} := n_{fasteners} \cdot n_{shearplanes} \cdot K_{ser} = 63.794 \, \frac{kN}{mm}$ (Final stiffness)

Connection stiffness
EC5-1-1, 7.1
Connection: Type 2 (T15)



$$\rho_{mean} := 430 \frac{kg}{m^3}$$ (Mean density)

$$d = 12\ mm$$ (Dowel diameter)

$$K_{ser} := 2 \cdot \frac{\rho_{mean}^{(1.5)} \cdot d}{23} \cdot \left(\frac{1}{\left(\frac{kg}{m^3}\right)^{(1.5)}} \right) \cdot \left(\frac{1}{m}\right) \frac{kN}{mm} = 9.304 \frac{kN}{mm}$$ (Stiffness per fastener per shear plane)

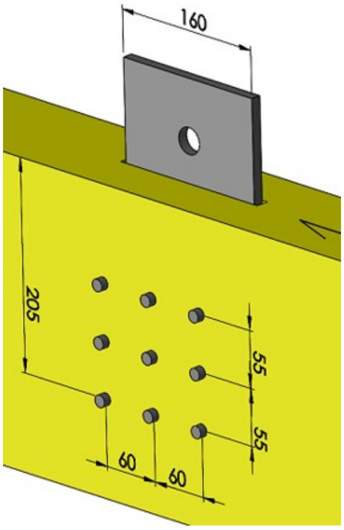
Final connection stiffness:

$$n_{fasteners} := 9$$ (Number of fasteners)

$$n_{shearplanes} := 2$$ (Number of shear planes)

$$K_{ser.final} := n_{fasteners} \cdot n_{shearplanes} \cdot K_{ser} = 167.478 \frac{kN}{mm}$$ (Final stiffness)

Connection stiffness
EC5-1-1, 7.1
Connection: Type 3 (T15)



$\rho_{mean} := 430\ \frac{kg}{m^3}$ (Mean density)

$d = 12\ mm$ (Dowel diameter)

$K_{ser} := 2 \cdot \frac{\rho_{mean}^{(1.5)} \cdot d}{23} \cdot \left(\frac{1}{\left(\frac{kg}{m^3} \right)^{(1.5)}} \right) \cdot \left(\frac{1}{m} \right) \frac{kN}{mm} = 9.304\ \frac{kN}{mm}$ (Stiffness per fastener per shear plane)

Final connection stiffness:

$n_{fasteners} := 9$ (Number of fasteners)

$n_{shearplanes} := 2$ (Number of shear planes)

$K_{ser.final} := n_{fasteners} \cdot n_{shearplanes} \cdot K_{ser} = 167.478\ \frac{kN}{mm}$ (Final stiffness)

Capacity check of anchoring plates

Minimum distances:
EC3-1-8, Table 3.3

Calculating both optimal and minimum distances.

$d := 40\ mm$ (Outer bolt diameter)

$d_0 := d + mm = 41\ mm$ (Bore holde diameter in steel plate)

$d_{low} := 25\ mm$ (Outer bolt diameter, lower part)

$d_{0.low} := d_{low} + 1\ mm = 26\ mm$ (Bore holde diameter in steel plate, lower part)

Optimal distances:

$e_{1.optimal} := 3 \cdot d_0 = 123\ mm$

$p_{1.optimal} := 3.75 \cdot d_0 = 153.75\ mm$

$e_{2.optimal} := 1.5 \cdot d_0 = 61.5\ mm$

$p_{2.optimal} := 3 \cdot d_0 = 123\ mm$

$e_{1.optimal.low} := 3 \cdot d_{0.low} = 78\ mm$

$p_{1.optimal.low} := 3.75 \cdot d_{0.low} = 97.5\ mm$

$e_{2.optimal.low} := 1.5 \cdot d_{0.low} = 39\ mm$

$p_{2.optimal.low} := 3 \cdot d_{0.low} = 78\ mm$

Minimum distances:

$e_{1.min} := 1.2 \cdot d_0 = 49.2\ mm$

$p_{1.min} := 2.2 \cdot d_0 = 90.2\ mm$

$e_{2.min} := 1.2 \cdot d_0 = 49.2\ mm$

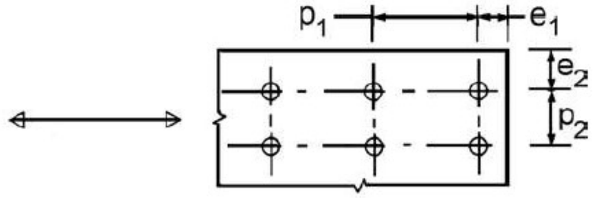
$p_{2.min} := 2.4 \cdot d_0 = 98.4\ mm$

$e_{1.min} := 1.2 \cdot d_{0.low} = 31.2\ mm$

$p_{1.min} := 2.2 \cdot d_{0.low} = 57.2\ mm$

$e_{2.min} := 1.2 \cdot d_{0.low} = 31.2\ mm$

$p_{2.min} := 2.4 \cdot d_{0.low} = 62.4\ mm$



Material properties

Due to geometrical restrictions on the hydraulick jack, the optimal distances may not be chosen. Controlling the capacity for the following chosen, relevant values.

$e_1 := 90\ mm$ (end distance in force direction)

$e_2 := 105\ mm$ (end distance perpendicular to force direction)

$e_{1.low} := 32\ mm$ (end distance in force direction, lower part)

$e_{2.low} := 45\ mm$ (end distance perpendicular to force direction, lower part)

$f_{ub} := 470\ \frac{N}{mm^2}$ (Ultimate stress of bolt)

$f_u := 470\ \frac{N}{mm^2}$ (Ulitmate stress of steel plate)

$t := 20\ mm$ (Thickness of steel plates)

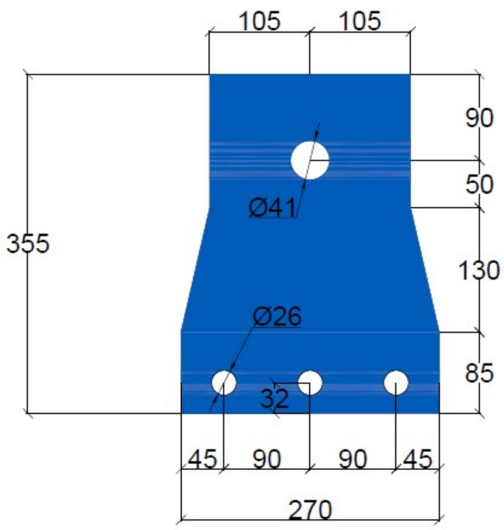
$b := 2 \cdot e_2 = 210\ mm$ (Width of steel plate)

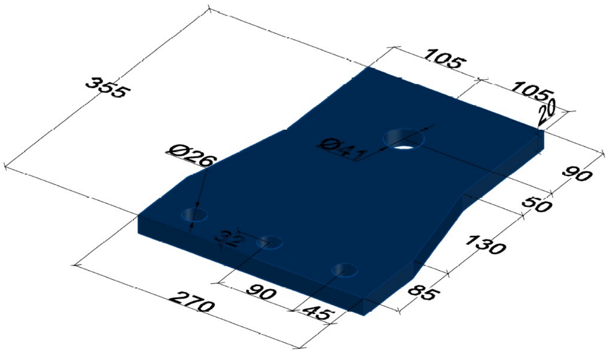
$b_{low} := 270\ mm$ (Width of steel plate, lower part)

$f_y := 355\ \frac{N}{mm^2}$ (Yielding stress of steel plate)

$\gamma_{M2} := 1.25$ (Material factor for steel in connections)

$\gamma_{M0} := 1.05$ (Material factor for steel)





Shear capacity of bolt:
EC3-1-8, Table 3.4
Upper part:

$\alpha_v := 0.6$

$d = 40\ mm$

$A := \frac{\pi}{4} \cdot d^2 = \left(1.257 \cdot 10^3\right)\ mm^2$

$n_{shear\_planes} := 2$

$F_{v.Rd} := n_{shear\_planes} \cdot \frac{\alpha_v \cdot f_{ub} \cdot A}{\gamma_{M2}} = 566.995\ kN$

Shear capacity of bolt:
EC3-1-8, Table 3.4
Lower part:

$\alpha_v := 0.6$

$d_{low} = 25\ mm$

$A_{low} := \frac{\pi}{4} \cdot {d_{low}}^2 = 490.874\ mm^2$

$n_{shear\_planes} := 2$

$n_{bolts} := 3$

$F_{v.Rd} := n_{bolts} \cdot n_{shear\_planes} \cdot \frac{\alpha_v \cdot f_{ub} \cdot A_{low}}{\gamma_{M2}} = 664.447\ kN$

Bearing capacity:
EC3-1-8, Table 3.4

Upper part:

$$\alpha_b := min\left(\frac{e_1}{3 \cdot d_0}, \frac{f_{ub}}{f_u}, 1.0\right) = 0.732$$

$$k_1 := min\left(2.8 \cdot \frac{e_2}{d_0} - 1.7, 2.5\right) = 2.5$$

$$F_{b.Rd} := \frac{k_1 \cdot \alpha_b \cdot f_u \cdot d \cdot t}{\gamma_{M2}} = 550.244 \ kN$$

Bearing capacity:
EC3-1-8, Table 3.4

Lower part:

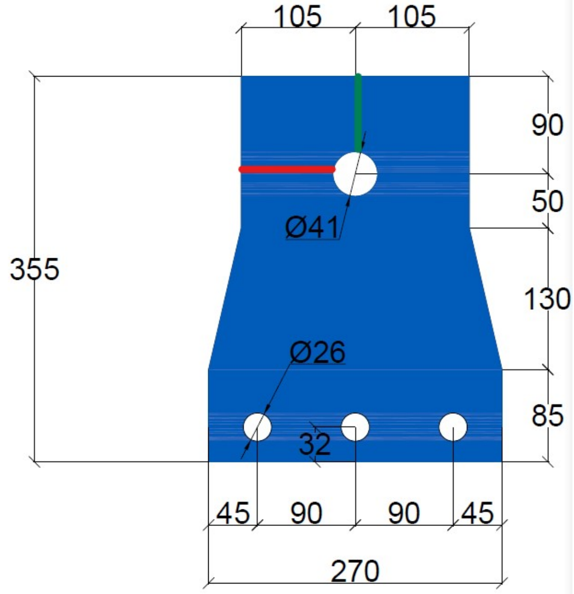
$$\alpha_b := min\left(\frac{e_{1.low}}{3 \cdot d_{0.low}}, \frac{f_{ub}}{f_u}, 1.0\right) = 0.41$$

$$k_1 := min\left(2.8 \cdot \frac{e_{2.low}}{d_{0.low}} - 1.7, 2.5\right) = 2.5$$

$$F_{b.Rd} := \frac{k_1 \cdot \alpha_b \cdot f_u \cdot d_{low} \cdot t}{\gamma_{M2}} = 192.821 \ kN$$

Comment: Assumed to be OK when taking friction into account.

Block tearing:
EC3-1-8, 3.10.2
Upper part:

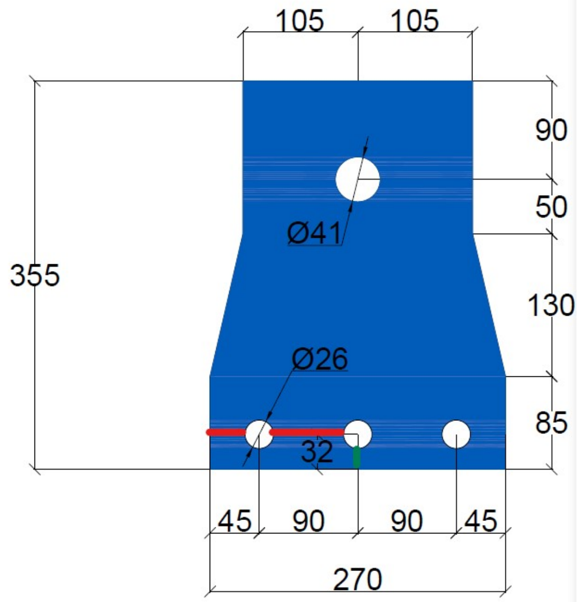


$$A_{nt} := \left(e_2 - \frac{d_0}{2}\right) \cdot t = \left(1.69 \cdot 10^3\right) \; mm^2$$

$$A_{nv} := \left(e_1 - \frac{d_0}{2}\right) \cdot t = \left(1.39 \cdot 10^3\right) \; mm^2$$

$$V_{eff.1.Rd} := \frac{f_u \cdot A_{nt}}{\gamma_{M2}} + \left(\frac{1}{\sqrt{3}}\right) \cdot \frac{f_y \cdot A_{nv}}{\gamma_{M0}} = 906.767 \; kN$$

Block tearing:
EC3-1-8, 3.10.2
Lower part:



$$p_{2.low} := 90 \; mm$$

$$A_{nt} := \left(e_{2.low} - \frac{d_{0.low}}{2} + p_{2.low} - d_{0.low}\right) \cdot t = \left(1.92 \cdot 10^3\right) \; mm^2$$

$$A_{nv} := \left(e_{1.low} - \frac{d_{0.low}}{2}\right) \cdot t = 380 \; mm^2$$

$$V_{eff.1.Rd} := \frac{f_u \cdot A_{nt}}{\gamma_{M2}} + \left(\frac{1}{\sqrt{3}}\right) \cdot \frac{f_y \cdot A_{nv}}{\gamma_{M0}} = 796.096 \; kN$$

Tension:
EC3-1-1, 3.10.2

Upper part:

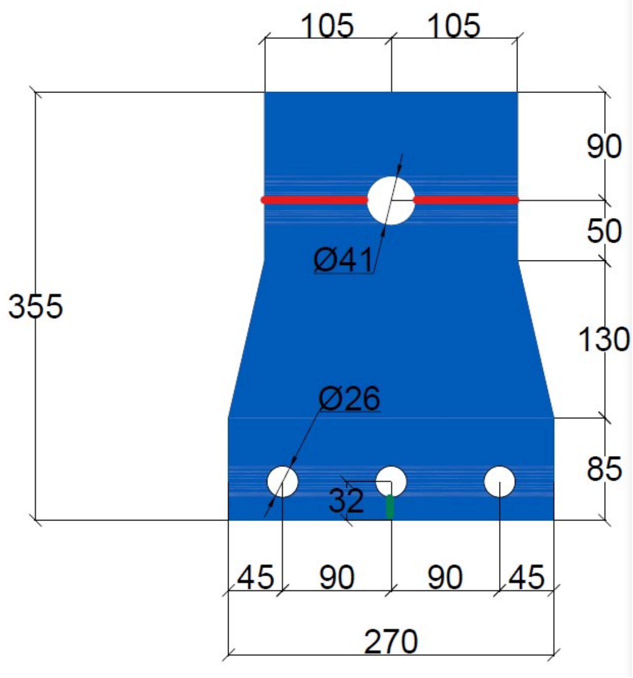
$b = 210\ mm$

$A := b \cdot t = (4.2 \cdot 10^3)\ mm^2$

$d_0 = 41\ mm$

$A_{net} := A - d_0 \cdot t = (3.38 \cdot 10^3)\ mm^2$

$$N_{t.Rd} := min\left(\frac{A \cdot f_y}{\gamma_{M0}}, \frac{0.9 \cdot A_{net} \cdot f_u}{\gamma_{M2}}\right) = (1.144 \cdot 10^3)\ kN$$



Tension:
EC3-1-1, 3.10.2

Lower part:

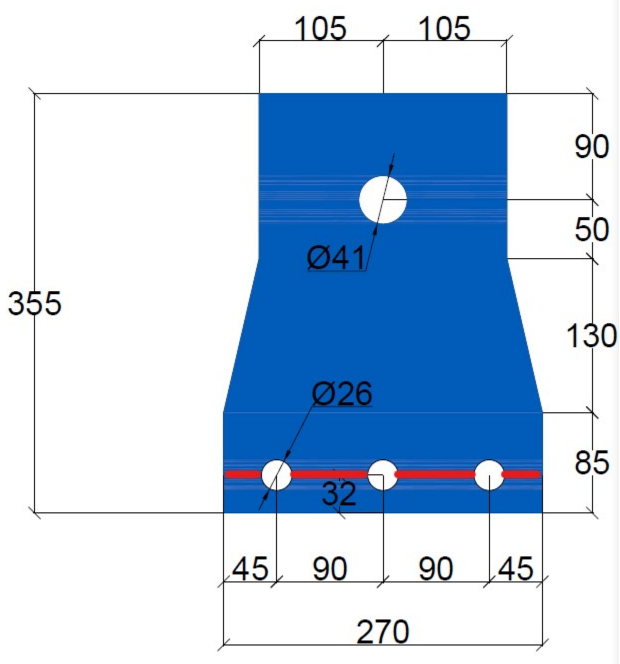
$b_{low} = 270\ mm$

$A := b_{low} \cdot t = (5.4 \cdot 10^3)\ mm^2$

$d_{0.low} = 26\ mm$

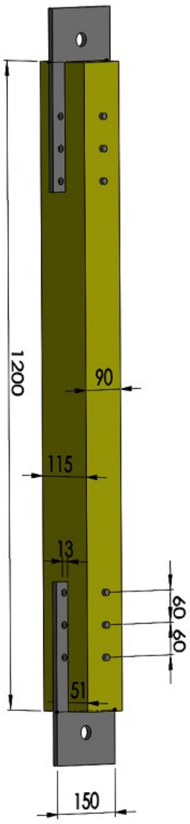
$A_{net} := A - 3 \cdot d_{0.low} \cdot t = (3.84 \cdot 10^3)\ mm^2$

$$N_{t.Rd} := min\left(\frac{A \cdot f_y}{\gamma_{M0}}, \frac{0.9 \cdot A_{net} \cdot f_u}{\gamma_{M2}}\right) = (1.299 \cdot 10^3)\ kN$$



Loading procedure: Elastic domain

Specimen Type 1



Estimated load capacity of the connection

$F_{est} := 55.6\ kN$

Loading frequency

$\omega := 0.1\ Hz$

Load levels

$F_{10} := 0.1 \cdot F_{est} = 5.56\ kN$ (Minimum loading)

$F_{20} := 0.2 \cdot F_{est} = 11.12\ kN$ (Middle load level)

$F_{30} := 0.3 \cdot F_{est} = 16.68\ kN$ (2. middle load level)

$F_{40} := 0.4 \cdot F_{est} = 22.24\ kN$ (Maximum loading)

Load phase 1: Cyclic in tension from 10 to 20% of F_{est}

$F_{m1} := \frac{F_{10} + F_{20}}{2} = 8.34\ kN$ (Mean load level)

$F_{a1} := F_{20} - F_{m1} = 2.78\ kN$ (Load amplitude)

Load phase 2: Cyclic in tension from 20 to 30% of F_{est}

$$F_{m2} := \frac{F_{20} + F_{30}}{2} = 13.9\ kN$$ (Mean load level)

$$F_{a2} := F_{30} - F_{m2} = 2.78\ kN$$ (Load amplitude)

Load phase 3: Cyclic in tension from 30 to 40% of F_{est}

$$F_{m3} := \frac{F_{30} + F_{40}}{2} = 19.46\ kN$$ (Mean load level)

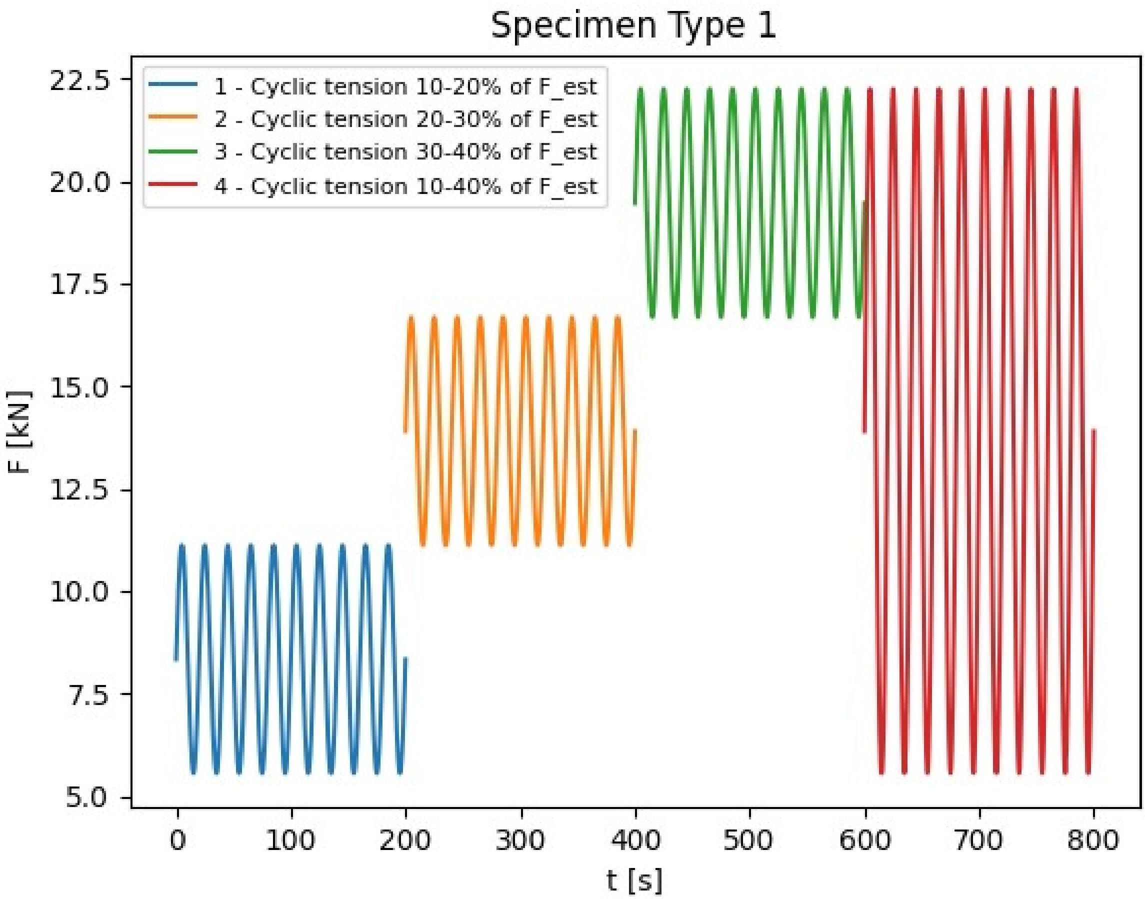
$$F_{a3} := F_{40} - F_{m3} = 2.78\ kN$$ (Load amplitude)

Load phase 4: Cyclic in tension from 10 to 40% of F_{est}

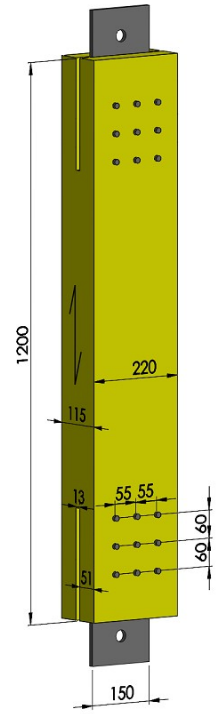
$$F_{m4} := \frac{F_{10} + F_{40}}{2} = 13.9\ kN$$ (Mean load level)

$$F_{a4} := F_{40} - F_{m4} = 8.34\ kN$$ (Load amplitude)

Estimate 10 cycles per load phase, to achieve stable results.



Loading procedure: Elastic domain



Specimen Type 2

Estimated load capacity of the connection

$F_{est} := 157\ kN$

Loading frequency

$\omega := 0.1\ Hz$

Sinusoidal load shape

Load levels

$F_{10} := 0.1 \cdot F_{est} = 15.7\ kN$ (Minimum loading)

$F_{20} := 0.2 \cdot F_{est} = 31.4\ kN$ (Middle load level)

$F_{30} := 0.3 \cdot F_{est} = 47.1\ kN$ (2. middle load level)

$F_{40} := 0.4 \cdot F_{est} = 62.8\ kN$ (Maximum loading)

Load phase 1: Cyclic in tension from 10 to 20% of F_{est}

$F_{m1} := \frac{F_{10} + F_{20}}{2} = 23.55\ kN$ (Mean load level)

$F_{a1} := F_{20} - F_{m1} = 7.85\ kN$ (Load amplitude)

Load phase 2: Cyclic in tension from 20 to 30% of F_{est}

$F_{m2} := \frac{F_{20} + F_{30}}{2} = 39.25\ kN$ (Mean load level)

$F_{a2} := F_{30} - F_{m2} = 7.85\ kN$ (Load amplitude)

Load phase 3: Cyclic in tension from 30 to 40% of F_{est}

$$F_{m3} := \frac{F_{30} + F_{40}}{2} = 54.95\ kN$$ (Mean load level)

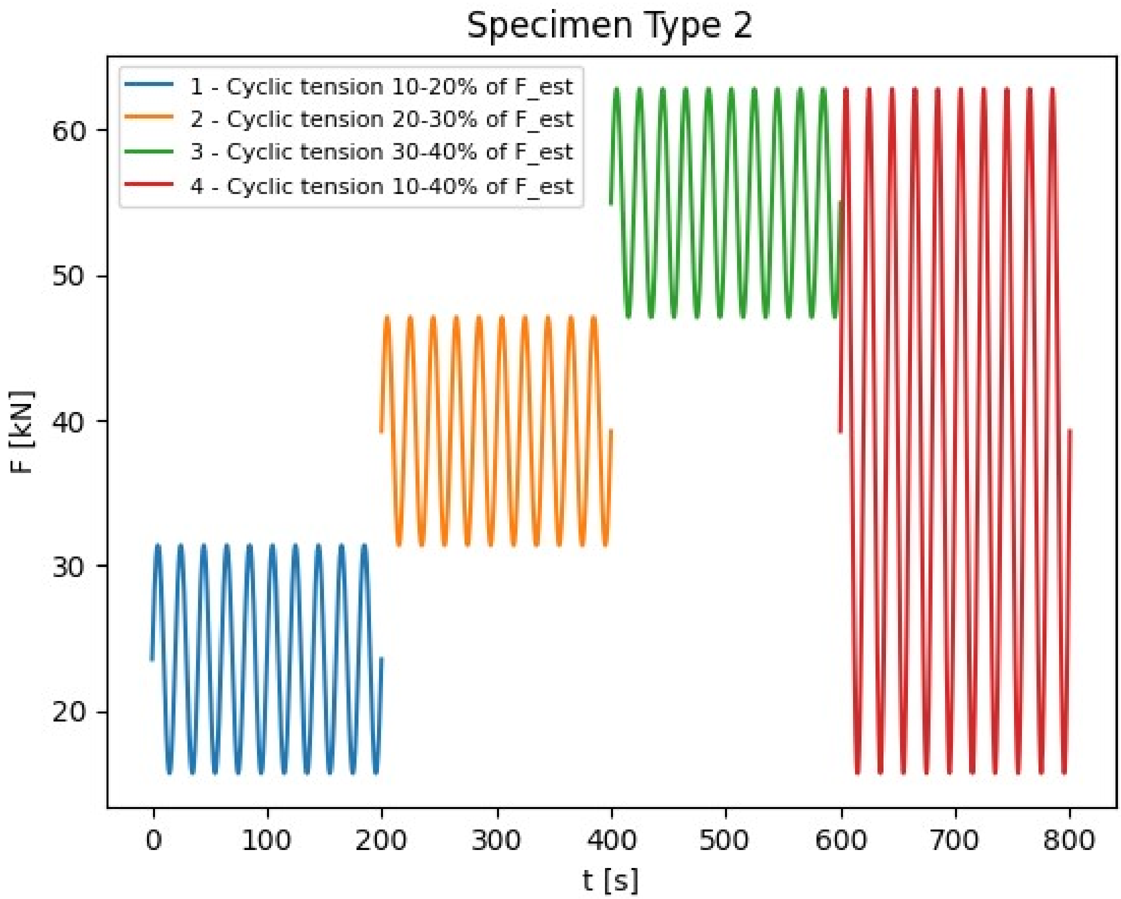
$$F_{a3} := F_{40} - F_{m3} = 7.85\ kN$$ (Load amplitude)

Load phase 4: Cyclic in tension from 10 to 40% of F_{est}

$$F_{m4} := \frac{F_{10} + F_{40}}{2} = 39.25\ kN$$ (Mean load level)

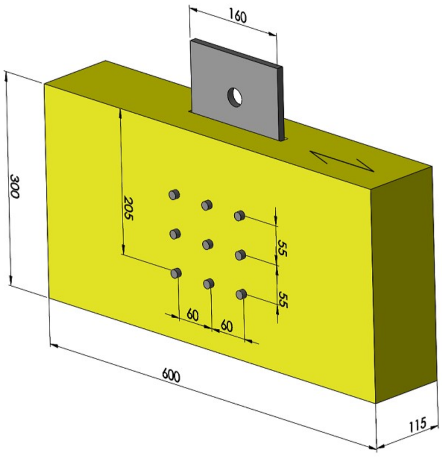
$$F_{a4} := F_{40} - F_{m4} = 23.55\ kN$$ (Load amplitude)

Estimate 10 cycles per load phase, to achieve stable results.



Loading procedure: Elastic domain

Specimen Type 3



Estimated load capacity of the connection

$F_{est} := 78\ kN$

Loading frequency

$\omega := 0.1\ Hz$

Sinusoidal load shape

Load levels

$F_{10} := 0.1 \cdot F_{est} = 7.8\ kN$ (Minimum loading)

$F_{20} := 0.2 \cdot F_{est} = 15.6\ kN$ (Middle load level)

$F_{30} := 0.3 \cdot F_{est} = 23.4\ kN$ (2. middle load level)

$F_{40} := 0.4 \cdot F_{est} = 31.2\ kN$ (Maximum loading)

Load phase 1: Cyclic in tension from 10 to 20% of F_{est}

$F_{m1} := \frac{F_{10} + F_{20}}{2} = 11.7\ kN$ (Mean load level)

$F_{a1} := F_{20} - F_{m1} = 3.9\ kN$ (Load amplitude)

Load phase 2: Cyclic in tension from 20 to 30% of F_{est}

$F_{m2} := \frac{F_{20} + F_{30}}{2} = 19.5\ kN$ (Mean load level)

$F_{a2} := F_{30} - F_{m2} = 3.9\ kN$ (Load amplitude)

Load phase 3: Cyclic in tension from 30 to 40% of F_{est}

$$F_{m3} := \frac{F_{30} + F_{40}}{2} = 27.3\ kN$$ (Mean load level)

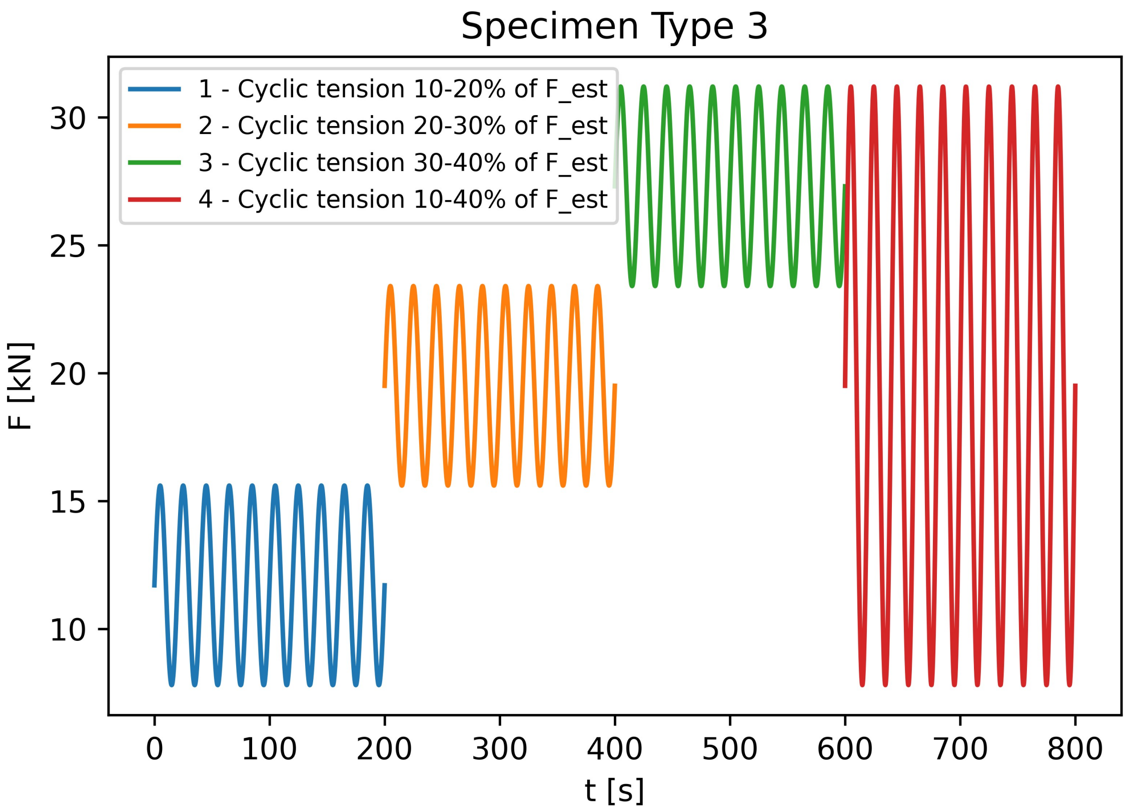
$$F_{a3} := F_{40} - F_{m3} = 3.9\ kN$$ (Load amplitude)

Load phase 4: Cyclic in tension from 10 to 40% of F_{est}

$$F_{m4} := \frac{F_{10} + F_{40}}{2} = 19.5\ kN$$ (Mean load level)

$$F_{a4} := F_{40} - F_{m4} = 11.7\ kN$$ (Load amplitude)

Estimate 10 cycles per load phase, to achieve stable results.



G Python codes/scripts

This appendix includes the Python script used to run the fully parametric Abaqus model. Comments are included in the script. The scripts used for data processing are included in the digital appendix only.

```python

# Timber part with dowel type connectors model for Abaqus/Python, Trygve Vangsnes, 06-2022

from abaqus import *
from abaqusConstants import *
from caeModules import *
import mesh
import os
import numpy as np
import scipy as sc
import math
import time
import sys
import matplotlib.pyplot as plt
import getpass
from scipy.optimize import curve_fit
session.journalOptions.setValues(replayGeometry=COORDINATE, recoverGeometry=COORDINATE)
DIR0 = os.path.abspath('')
TOL = 1e-6


def make_geometry(model,timberGeom,plGeom,meshSize,timberSetup,dowelModelling):
    """This function takes in the Abaqus model to generate geometry in. The dimensions of the timber part:
     b, h, l. The dowel diameter d and a list of the dowel setup distances: a1,a2,e1,e2, n_row and n_col.
     Steel plate thickness tPl, steel plate length tL and number of steel plates nPl.
     The geoemtry also takes in the mesh size as a variable as well as what timberSetup is to be tested, and how the dowel is to be modelled."""

    # # Unpack the timber-geometry
    b,h,l,d,a1,a2,e1,e2 = timberGeom[0],timberGeom[1],timberGeom[2],timberGeom[3],timberGeom[4],timberGeom[5],timberGeom[6],timberGeom[7]
    tRing1, tRing2,n_row,n_col,dAngle = timberGeom[8],timberGeom[9],timberGeom[10],timberGeom[11],timberGeom[12]
    plThickness,plLength,plLOut,plWidth,a1Pl,a2Pl,e1Pl,e2Pl,plGap,plCutOut = plGeom[0],plGeom[1],plGeom[2],plGeom[3],plGeom[4],plGeom[5],plGeom[6],plGeom[7],plGeom[8],plGeom[9]


    # Do some easy checks to make sure the input is correct
    if (a2*(n_col-1)+2*e2) > h:
        raise ValueError('Not space for that many dowels with the given spacing')
    if n_row and n_col != 1:
        if (a2Pl*(n_col-1)+2*e2Pl) > plWidth:
            raise ValueError('Plate dimensions are wrong')
    if d/2 >= a2/2:
        raise ValueError('To large dowel diameter')
    if dowelModelling not in('0Ring','1Ring','2Ring'):
        raise ValueError('Check if correct dowelmodelling technique is entered')

    # Define some useful parameters for geometry generation.
    bcx = (h-n_col*a2)/2            # Bottom left corner x-coordinates
    bcy = e1-a1/2                   # Bottom left corner y-coordinates
    print(dowelModelling)
    print(timberSetup)
    # draw the sketch for  a dowel-zone
    s = model.ConstrainedSketch(name = 'timberPart', sheetSize = 200.0)
    s.rectangle(point1 = (bcx, bcy), point2 = (bcx+a2, bcy+a1))
    if dowelModelling == '0Ring':
        dowelZoneCutout = (d/2)
    elif dowelModelling == '1Ring':
        dowelZoneCutout = (tRing1+d/2)
    elif dowelModelling == '2Ring':
        dowelZoneCutout = (tRing1+tRing2+d/2)
    s.CircleByCenterPerimeter(center = (bcx+a2/2, bcy+a1/2), point1 = (bcx+a2/2-dowelZoneCutout, bcy+a1/2))

    # draw the sketch for the outer part of the timber, with the dowelzone as cut-out
    t = model.ConstrainedSketch(name = 'outerTimberPart', sheetSize = 200.0)
    t.rectangle(point1 = (0.0  , 0.0),  point2 = (h , l))
    t.rectangle(point1 = (bcx  , bcy),  point2 = (bcx+n_col*a2  , bcy+n_row*a1))

    # draw the dowel sketch
    ds = model.ConstrainedSketch(name = 'dowel', sheetSize = 200.0)
    ds.CircleByCenterPerimeter(center = (bcx+a2/2, bcy+a1/2), point1 = (bcx+a2/2-(d/2), bcy+a1/2))

    # depending on the chosen dowelzone modelling sketches for either 1 or 2 rings are drawn
    if dowelModelling in ('1Ring','2Ring'):
        r1 = model.ConstrainedSketch(name = 'InnerRing', sheetSize = 200.0)
        r1.CircleByCenterPerimeter(center = (bcx+a2/2, bcy+a1/2), point1 = (bcx+a2/2-(d/2), bcy+a1/2))
        r1.CircleByCenterPerimeter(center = (bcx+a2/2, bcy+a1/2), point1 = (bcx+a2/2-(tRing1+d/2), bcy+a1/2))
        if dowelModelling == '2Ring':
            r2 = model.ConstrainedSketch(name = 'OuterRing', sheetSize = 200.0)
            r2.CircleByCenterPerimeter(center = (bcx+a2/2, bcy+a1/2), point1 = (bcx+a2/2-(d/2+tRing1), bcy+a1/2))
            r2.CircleByCenterPerimeter(center = (bcx+a2/2, bcy+a1/2), point1 = (bcx+a2/2-(tRing1+tRing2+d/2), bcy+a1/2))

    # sketch the plate, and make cutouts for the dowels
    pl = model.ConstrainedSketch(name = 'steelPlate', sheetSize = 200.0)
    pl.rectangle( point1 = (0.0 , -plLOut), point2 = (plWidth,plLength) )

    if (n_col % 2) == 0:
        colCounter = int(n_col/2)
    else:
```

```
        colCounter = int(np.ceil(n_col/2))+1
    for i in range(n_row):
        for j in range(colCounter):
            if (n_col % 2) == 0: #check if number of columns is even
                xCoordL = plWidth/2 - a2Pl/2 - j*a2Pl
                xCoordR = plWidth/2 + a2Pl/2 + j*a2Pl
                yCoord = e1 + i*a1Pl
                pl.CircleByCenterPerimeter(center = (xCoordL, yCoord), point1 = (xCoordL-d/2, yCoord))
                pl.CircleByCenterPerimeter(center = (xCoordR, yCoord), point1 = (xCoordR-d/2, yCoord))

            else: # for the odd number of columns
                if j==0:
                    xCoord = plWidth/2
                    yCoord = e1 + i*a1Pl#e1Pl + i*a1Pl
                    pl.CircleByCenterPerimeter(center = (xCoord, yCoord), point1 = (xCoord-d/2, yCoord))

                else:
                    xCoordL = plWidth/2 - j*a2Pl
                    xCoordR = plWidth/2 + j*a2Pl
                    yCoord  = e1 + i*a1Pl#e1Pl + i*a1Pl
                    pl.CircleByCenterPerimeter(center = (xCoordL, yCoord), point1 = (xCoordL-d/2, yCoord))
                    pl.CircleByCenterPerimeter(center = (xCoordR, yCoord), point1 = (xCoordR-d/2, yCoord))

    bsym = b+plGap+plThickness/2
    # create the 3-dimensional parts
    tp = model.Part(dimensionality = THREE_D, name = 'timberPart', type = DEFORMABLE_BODY)
    tp.BaseSolidExtrude(depth = b, sketch = s)

    oTp = model.Part(dimensionality = THREE_D, name = 'outerTimberPart', type = DEFORMABLE_BODY)
    oTp.BaseSolidExtrude(depth = bsym, sketch = t)

    dowel = model.Part(dimensionality = THREE_D, name = 'dowels', type = DEFORMABLE_BODY)
    dowel.BaseSolidExtrude(depth = bsym, sketch = ds) #(depth = b+(tPl+1)/2, sketch = ds)

    if dowelModelling in ('1Ring','2Ring'):
        ring1 = model.Part(dimensionality = THREE_D, name = 'InnerRing', type = DEFORMABLE_BODY)
        ring1.BaseSolidExtrude(depth = b, sketch = r1)
        if dowelModelling == '2Ring':
            ring2 = model.Part(dimensionality = THREE_D, name = 'OuterRing', type = DEFORMABLE_BODY)
            ring2.BaseSolidExtrude(depth = b, sketch = r2)

    steelPl = model.Part(dimensionality = THREE_D, name = 'steelPlate', type = DEFORMABLE_BODY)
    steelPl.BaseSolidExtrude(depth = plThickness/2, sketch = pl)

    # # create sets, surfaces, and reference point
    dowel.Set(   name = 'dowelEnd',    faces = dowel.faces.getByBoundingBox(  zMin = b+plGap+plThickness/2-TOL, zMax=b+plGap+plThickness/2+TOL )) #dowel.faces.getByBoundingBox(zMin =b+(tPl+1)/2-TOL, zMax=b+(tPl+1)/2+TOL ))
    oTp.Set(     name = 'top'      ,    faces = oTp.faces.getByBoundingBox(    yMin = float(l)-TOL))
    if timberSetup == 'S3':
        oTp.Set( name = 'S3fix',        faces = oTp.faces.findAt(((0.0, l/2, b/2), ), ((h, l/2, b/2), ), ))
    steelPl.Set( name = 'symPlane',    faces = steelPl.faces.getByBoundingBox(zMin = plThickness/4))
    steelPl.Set( name = 'plateEnd',    faces = steelPl.faces.getByBoundingBox(yMax = -plLOut+TOL))
    steelPl.Set( name = 'plateTop',    faces = steelPl.faces.getByBoundingBox(yMin = plLength-TOL))
    steelPl.Surface(name='plateEnd2',  side1Faces = steelPl.faces.getByBoundingBox(yMax = -plLOut+TOL))
    steelPl.Set( name = 'plateEdge',   edges = steelPl.edges.findAt(((plWidth/2, -plLOut, plThickness/2), ), ((plWidth/2, plLength, plThickness/2), ), ((0.0, 0.0, plThickness/2), ), ((plWidth, 0.0, plThickness/2), ),))
    oTp.Set(     name =  'entire',     cells = oTp.cells[:])
    tp.Set(      name =  'entire',     cells = tp.cells[:])
    dowel.Set(   name =  'entire',     cells = dowel.cells[:])
    if dowelModelling in ('1Ring','2Ring'):
        ring1.Set(   name =  'entire',     cells = ring1.cells[:])
        if dowelModelling == '2Ring':
            ring2.Set(   name =  'entire',     cells = ring2.cells[:])
    steelPl.Set( name =  'entire',     cells = steelPl.cells[:])
    if dowelModelling in ('1Ring','2Ring'):
        tp.Surface(   name='timberIN'     , side1Faces=tp.faces.findAt(((    bcx+a2/2-dowelZoneCutout, bcy+a1/2     ,b/2), )))


    ## Create cut out for steel-plates, need sketch for the cut-out first
    sidePlane = oTp.faces.findAt((0.0, l/2, bsym/2), )
    sideEdge = oTp.edges.findAt((0.0,l/2, bsym), )
    sP = model.ConstrainedSketch(name         ='steelPlateSym', sheetSize = 2*l, transform = oTp.MakeSketchTransform(
                               sketchPlane  = sidePlane   , sketchPlaneSide = SIDE1,
                               sketchUpEdge = sideEdge    , sketchOrientation =RIGHT, origin = (0.0, 0.0, bsym)))
    oTp.projectReferencesOntoSketch(filter = COPLANAR_EDGES, sketch = sP)
    sP.rectangle(point1 = (0, 0), point2 = (-plThickness/2-plGap, plCutOut))#bcy+n_row*a1))
    oTp.CutExtrude(flipExtrudeDirection = OFF, sketch = sP, sketchOrientation = RIGHT, sketchPlane = sidePlane, sketchPlaneSide = SIDE1, sketchUpEdge = sideEdge)

    # create datums to make nice mesh on steel-plates
    # One datum per row and one per plane
    for i in range(n_row):
        yC = e1 + i*a1Pl
        yCOver  = e1 + i*a1Pl + 1.5*d
        yCUnder = e1 + i*a1Pl - 1.5*d
        xC = 0.0
        zC = 0.0
        plDatum = steelPl.DatumPlaneByThreePoints(((xC,yC,zC)),((xC,yC,zC+b)),((xC+plWidth,yC,zC)))
        steelPl.PartitionCellByDatumPlane(cells=steelPl.cells[:],datumPlane = steelPl.datums[plDatum.id ])
```

```
        plDatum1 = steelPl.DatumPlaneByThreePoints(((xC,yCover,zC)),((xC,yCover,zC+b)),((xC+plWidth,yCover,zC)))
        steelPl.PartitionCellByDatumPlane(cells=steelPl.cells[:],datumPlane = steelPl.datums[plDatum1.id ])
        plDatum2 = steelPl.DatumPlaneByThreePoints(((xC,yCUnder,zC)),((xC,yCUnder,zC+b)),((xC+plWidth,yCUnder,zC)))
        steelPl.PartitionCellByDatumPlane(cells=steelPl.cells[:],datumPlane = steelPl.datums[plDatum2.id ])

    # Pick and create the surfaces to connect the dowel to the steel-plate by a model-tie.
    ## Need fix for two columns in S2 test-specimen
    for i in range(n_row):
        yCoord = e1 + i*a1Pl+d/2
        for j in range(n_col):
            if n_col==1:
                xCoord = plWidth/2
            else:
                xCoord = e2Pl+j*a2Pl
            steelPl.Surface(name='dowelTiePl_'+str(i+1)+'-'+str(j+1), side1Faces=steelPl.faces.findAt(((xCoord,yCoord,plThickness/4),)))

    # create datums and partition to make nice mesh on steel-plates also for the columns so that all circles are split into 4

    plDatums = {}
    for j in range(n_col):
        yC,zC = 0.0, 0.0
        if (n_col % 2) == 0: #check if number of columns is even
            xC = plWidth/2-colCounter*a2Pl/2+a2Pl*j
        elif n_col != 1:
            xC = e2Pl+j*a2Pl
        elif n_col == 1:
            xC = plWidth/2
        xcVec = [xC, xC-1.5*d, xC+1.5*d ]
        for count, xC in enumerate(xcVec):
            plDatums['plDat-'+str(j+1)+'-'+str(count+1)] = steelPl.DatumPlaneByThreePoints(((xC,yC,zC)),((xC,yC,zC+b)),((xC,yC+b,zC)))
            steelPl.PartitionCellByDatumPlane(cells=steelPl.cells[:],datumPlane = steelPl.datums[plDatums['plDat-'+str(j+1)+'-'+str(count+1)].id ])

    # Choose if the dowel connects to the timberpart or ring1
    if dowelModelling in ('1Ring','2Ring'):
        partToConnect = ring1
    else:
        partToConnect = tp

    #Make surfaces to make tie constraint between the parts
    if dowelModelling in ('1Ring','2Ring'):
        partToConnect.Surface(name='innerRingOUT' , side1Faces=partToConnect.faces.findAt((( bcx+a2/2-(tRing1+d/2)       , bcy+a1/2 ,b/2), )))

    # partition dowel to connect one part to the wood and one part of it to the steelplate
    timberEndDat = dowel.DatumPlaneByThreePoints(((0.0,0.0,b)),((plWidth,0.0,b)),((0.0,plLength,b)))
    dowel.PartitionCellByDatumPlane(cells=dowel.cells[:],datumPlane = dowel.datums[timberEndDat.id ])
    timberCutDat = oTp.DatumPlaneByThreePoints(((0.0,0.0,b)),((plWidth,0.0,b)),((0.0,plLength,b)))
    oTp.PartitionCellByDatumPlane(cells=oTp.cells[:],datumPlane = oTp.datums[timberCutDat.id ])
    steelStartDat = dowel.DatumPlaneByThreePoints(((0.0,0.0,b+plGap)),((plWidth,0.0,b+plGap)),((0.0,plLength,b+plGap)))
    dowel.PartitionCellByDatumPlane(cells=dowel.cells[:],datumPlane = dowel.datums[steelStartDat.id ])

    # Create variable angle for the size of the surface tie between dowel and region 1
    # do this by creating new partition in symmetric shape. Also pick surfaces on the dowel and ring1 to use in tie-constraint
    # Accept input in the form of angles every 22.5 degree from 22.5 to 180.
    # Split the half circle into 2, 4, 8 or 16 parts in order to achieve this.


    dowelDatums = {}
    partToConnectDatums = {}
    if dAngle == 22.5 or dAngle == 45.0 or dAngle == 90.0 or dAngle == 180.0:
        n_div  = int(float(360)/float(dAngle))
        dTheta = np.deg2rad(float(180)/float(n_div))
        angleList = np.arange(0,math.pi,dTheta)
        firstAngles = [0.0, math.pi/2]
        for i,theta_i in enumerate(firstAngles):
            dowelDatums["doDat{0}".format(i)] = dowel.DatumPlaneByThreePoints(((bcx+a2/2,bcy+a1/2,0.0)),((bcx+a2/2,bcy+a1/2,b)),((bcx+a2/2 + d/2*np.cos(theta_i)    , bcy+a1/2 + d/2*np.sin(theta_i) ,0.0)))
            dowel.PartitionCellByDatumPlane(cells=dowel.cells[:],datumPlane = dowel.datums[dowelDatums["doDat{0}".format(i)].id ])

            partToConnectDatums["pcDat{0}".format(i)] = partToConnect.DatumPlaneByThreePoints(((bcx+a2/2,bcy+a1/2,0.0)),((bcx+a2/2,bcy+a1/2,b)),((bcx+a2/2 + d/2*np.cos(theta_i)    , bcy+a1/2 + d/2*np.sin(theta_i) ,0.0)))
            partToConnect.PartitionCellByDatumPlane(cells=partToConnect.cells[:],datumPlane = partToConnect.datums[partToConnectDatums["pcDat{0}".format(i)].id ])
            if i==0:
                dowel.Surface(name='dowelSurfPl'  , side1Faces=dowel.faces.findAt(((bcx+a2/2 + d/2*np.cos(theta_i)    , bcy+a1/2 + d/2*np.sin(theta_i) ,b+plGap+plThickness/4), ), ))

        newAngleList = np.setdiff1d(angleList,firstAngles)
        for i,theta_i in enumerate(newAngleList):
            dowelDatums["doDat{0}".format(i)] = dowel.DatumPlaneByThreePoints(((bcx+a2/2,bcy+a1/2,0.0)),((bcx+a2/2,bcy+a1/2,b)),((bcx+a2/2 + d/2*np.cos(theta_i)    , bcy+a1/2 + d/2*np.sin(theta_i) ,0.0)))
            dowel.PartitionCellByDatumPlane(cells=dowel.cells[:],datumPlane = dowel.datums[dowelDatums["doDat{0}".format(i)].id ])

            partToConnectDatums["pcDat{0}".format(i)] = partToConnect.DatumPlaneByThreePoints(((bcx+a2/2,bcy+a1/2,0.0)),((bcx+a2/2,bcy+a1/2,b)),((bcx+a2/2 + d/2*np.cos(theta_i)    , bcy+a1/2 + d/2*np.sin(theta_i) ,0.0)))
            partToConnect.PartitionCellByDatumPlane(cells=partToConnect.cells[:],datumPlane = partToConnect.datums[partToConnectDatums["pcDat{0}".format(i)].id ])

```

```
        dowel.Surface(name='dowelSurf'    , side1Faces=dowel.faces.findAt(((bcx+a2/2 + d/2*np.cos((3*math.pi)/2-TOL)   , bcy+a1/2 + d/2*np.sin((3*math.pi)/2-TOL) ,b/2), ), ((bcx+a2/2 + d/2*np.cos((3*math.pi)/2+TOL)   , bcy+a1/2 + d/2*np.sin((3*math.pi)/2+TOL) ,b/2), ), ))
        partToConnect.Surface(name='innerRingIN'  , side1Faces=partToConnect.faces.findAt(((bcx+a2/2 + d/2*np.cos((3*math.pi)/2-TOL) , bcy+a1/2 + d/2*np.sin((3*math.pi)/2-TOL) ,b/2), ), ((bcx+a2/2 + d/2*np.cos((3*math.pi)/2+TOL)   , bcy+a1/2 + d/2*np.sin((3*math.pi)/2+TOL) ,b/2), ), ))
    else:
        n_div  = 16.0
        dTheta = np.deg2rad(float(180)/float(n_div))
        dAngleR = np.deg2rad(dAngle)
        angleList = np.arange(0,math.pi,dTheta)
        firstAngles = [0.0, (math.pi-dAngleR)/2, (math.pi-dAngleR)/2+dAngleR]
        for i,theta_i in enumerate(firstAngles):
            dowelDatums["doDat{0}".format(i)] = dowel.DatumPlaneByThreePoints(((bcx+a2/2,bcy+a1/2,0.0)),((bcx+a2/2,bcy+a1/2,b)), ((bcx+a2/2 + d/2*np.cos(theta_i)    , bcy+a1/2 + d/2*np.sin(theta_i) ,0.0)))
            dowel.PartitionCellByDatumPlane(cells=dowel.cells[:],datumPlane = dowel.datums[dowelDatums["doDat{0}".format(i)].id ])

            partToConnectDatums["pcDat{0}".format(i)] = partToConnect.DatumPlaneByThreePoints(((bcx+a2/2,bcy+a1/2,0.0)), ((bcx+a2/2,bcy+a1/2,b)),((bcx+a2/2 + d/2*np.cos(theta_i)    , bcy+a1/2 + d/2*np.sin(theta_i) ,0.0)))
            partToConnect.PartitionCellByDatumPlane(cells=partToConnect.cells[:],datumPlane = partToConnect.datums[partToConnectDatums["pcDat{0}".format(i)].id ])
            if i==0:
                dowel.Surface(name='dowelSurfPl'  , side1Faces=dowel.faces.findAt(((bcx+a2/2 + d/2*np.cos(theta_i)   , bcy+a1/2 + d/2*np.sin(theta_i) ,b+plGap+plThickness/4), ), ))
        dowel.Surface(name='dowelSurf'    , side1Faces=dowel.faces.findAt(((bcx+a2/2 + d/2*np.cos((3*math.pi)/2-TOL)   , bcy+a1/2 + d/2*np.sin((3*math.pi)/2-TOL) ,b/2), ),  ))
        partToConnect.Surface(name='innerRingIN'  , side1Faces=partToConnect.faces.findAt(((bcx+a2/2 + d/2*np.cos((3*math.pi)/2-TOL) , bcy+a1/2 + d/2*np.sin((3*math.pi)/2-TOL) ,b/2), ),  ))

        newAngleList = np.setdiff1d(angleList,firstAngles)
        for i,theta_i in enumerate(newAngleList):
            dowelDatums["doDat{0}".format(i)] = dowel.DatumPlaneByThreePoints(((bcx+a2/2,bcy+a1/2,0.0)),((bcx+a2/2,bcy+a1/2,b)), ((bcx+a2/2 + d/2*np.cos(theta_i)    , bcy+a1/2 + d/2*np.sin(theta_i) ,0.0)))
            dowel.PartitionCellByDatumPlane(cells=dowel.cells[:],datumPlane = dowel.datums[dowelDatums["doDat{0}".format(i)].id ])

            partToConnectDatums["pcDat{0}".format(i)] = partToConnect.DatumPlaneByThreePoints(((bcx+a2/2,bcy+a1/2,0.0)), ((bcx+a2/2,bcy+a1/2,b)),((bcx+a2/2 + d/2*np.cos(theta_i)    , bcy+a1/2 + d/2*np.sin(theta_i) ,0.0)))
            partToConnect.PartitionCellByDatumPlane(cells=partToConnect.cells[:],datumPlane = partToConnect.datums[partToConnectDatums["pcDat{0}".format(i)].id ])
    if dowelModelling == '2Ring':
        ring2.Surface(name='outerRingIN'  , side1Faces=ring2.faces.findAt((( bcx+a2/2-(tRing1+d/2)       , bcy+a1/2     ,b/2), )))
        ring2.Surface(name='outerRingOUT' , side1Faces=ring2.faces.findAt((( bcx+a2/2-(tRing1+tRing2+d/2), bcy+a1/2     ,b/2), )))

    # create surfaces on the timber-part used to model the dowel-zone to connect in the tie-commands
    tp.Surface(   name='timberOUT_L'    , side1Faces=tp.faces.getByBoundingBox( xMin = bcx-TOL, xMax = bcx+TOL))
    tp.Surface(   name='timberOUT_R'    , side1Faces=tp.faces.getByBoundingBox( xMin = bcx+a2-TOL, xMax = bcx+a2+TOL))
    tp.Surface(   name='timberOUT_B'    , side1Faces=tp.faces.getByBoundingBox( yMin = bcy-TOL, yMax = bcy+TOL))
    tp.Surface(   name='timberOUT_T'    , side1Faces=tp.faces.getByBoundingBox( yMin = bcy+a1-TOL, yMax = bcy+a1+TOL))

    # partition the inner edges of the outer timber part to match with the "building blocks" in the tie command
    if n_row != 3:
        datSymXZ = oTp.DatumPlaneByThreePoints(((0.0,plCutOut,0.0)),((bcx,plCutOut,0.0)),((bcx,plCutOut,b)))
        oTp.PartitionCellByDatumPlane(cells=oTp.cells[:],datumPlane = oTp.datums[datSymXZ.id])

    datXZtemp=[]
    for j in range(n_row+1):
        datXZtemp.append(oTp.DatumPlaneByThreePoints(((0.0,bcy+a1*j,0.0)),((bcx,bcy+a1*j,0.0)),((bcx,bcy+a1*j,b))) )
        oTp.PartitionCellByDatumPlane(cells=oTp.cells[:],datumPlane = oTp.datums[datXZtemp[j].id])
    for i in range(len(datXZtemp)-1):
        oTp.Surface(  name='outerTimberIN-L'+str(i+1), side1Faces=oTp.faces.findAt(((   bcx           , bcy+a1/2+a1*i ,b/2), )))
        oTp.Surface(  name='outerTimberIN-R'+str(i+1), side1Faces=oTp.faces.findAt(((   bcx+a2*n_col , bcy+a1/2+a1*i ,b/2), )))

    oTp.Set(faces= oTp.faces.findAt(((h/2, l-3*TOL, bsym), )), name='sym-midplane-BC')
    if timberSetup == 'S3':
        holdDownWidth = 100
        xC = holdDownWidth
        xM = holdDownWidth/2
        for i in range(2):
            datHoldDown = oTp.DatumPlaneByThreePoints(((xC,l,b)),((xC,0.0,0.0)),((xC,l,0.0)))
            oTp.PartitionCellByDatumPlane(cells=oTp.cells[:],datumPlane = oTp.datums[datHoldDown.id])
            oTp.Set( name = 'holdDown'+str(i+1) ,faces = oTp.faces.findAt(((xM,0,b/2), ),))# ((xM,0,bsym-2*TOL),),))
            xC += h-2*holdDownWidth
            xM = xC+holdDownWidth/2

    datZYtemp=[]
    for i in range(n_col+1):
        datZYtemp.append(oTp.DatumPlaneByThreePoints(((bcx+a2*i,bcy,0.0)),((bcx+a2*i,bcy,b/2)),((bcx+a2*i,bcy+a1,0.0))) )
        oTp.PartitionCellByDatumPlane(cells=oTp.cells[:],datumPlane = oTp.datums[datZYtemp[i].id])
    for j in range(len(datZYtemp)-1):
        oTp.Surface(  name='outerTimberIN-T'+str(j+1), side1Faces=oTp.faces.findAt((( bcx+a2/2+a2*j , bcy + n_row*a1 , b/2), )))
        oTp.Surface(  name='outerTimberIN-B'+str(j+1), side1Faces=oTp.faces.findAt((( bcx+a2/2+a2*j , bcy             , b/2), )))

    # Create a node set on top of edges of steel plates to use for relative displacment measurement, like LVDT measurement in lab.
    steelPl.Set( name = 'plateTop2',   faces = steelPl.faces.findAt(((2*TOL, plLength, plThickness/4), ), ((plWidth-2*TOL, plLength, plThickness/4), ), ))

    # create a reference-point and a set at the bottom of the steel-plate to gather reaction forces, RF2, in the Force-displacement testing
    rp = steelPl.ReferencePoint(point=(plWidth/2, -plLOut,plThickness/2))
    steelPl.Set(name='SteelPL_Bot', referencePoints=(steelPl.referencePoints[rp.id], ))
```

```python
    # if the timberSetup is S3, the displacement must be measured as in lab subtracting deflection at the LVDT placement
    if timberSetup == 'S3':
        rp = oTp.ReferencePoint(point=(h/2,l,b/2))
        oTp.Set(name='S3-disp',referencePoints=(oTp.referencePoints[rp.id], ) )
        oTp.Surface(name='S3-disp',side1Faces = oTp.faces.findAt(((h/2,l,b/2), ),) )

    # # Meshing the parts
    # Mesh sizes for the different regions
    oTpMesh, tpMesh, dowelMesh, ring1Mesh,ring2Mesh,stPlMesh   = meshSize[0], meshSize[1], meshSize[2], meshSize[3], meshSize[4],
meshSize[5]

    # Assign mesh for the differnet parts
    # Create partitions on the parts that have not been partioned yet to achieve smooth mesh
    # Choose what type of elements that are used in the model: linear 8 node reduced integration,
    # linear 8 node or quadratic 20 node reduced integration
    element_types = {
    'linR' : (mesh.ElemType(elemCode=C3D8R, elemLibrary=STANDARD, secondOrderAccuracy=OFF,
            kinematicSplit=AVERAGE_STRAIN, hourglassControl=DEFAULT,
            distortionControl=DEFAULT), mesh.ElemType(elemCode=C3D6, elemLibrary=STANDARD),
            mesh.ElemType(elemCode=C3D4, elemLibrary=STANDARD)),
    'lin'  : (mesh.ElemType(elemCode=C3D8, elemLibrary=STANDARD, secondOrderAccuracy=OFF,
            distortionControl=DEFAULT), mesh.ElemType(elemCode=C3D6, elemLibrary=STANDARD),
            mesh.ElemType(elemCode=C3D4, elemLibrary=STANDARD )),
    'quad' : (mesh.ElemType(
            elemCode=C3D20R, elemLibrary=STANDARD), mesh.ElemType(elemCode=C3D15,
            elemLibrary=STANDARD), mesh.ElemType(elemCode=C3D10, elemLibrary=STANDARD))
    }
    element_type = element_types['linR']

    oTp.setMeshControls(elemShape=HEX, regions=oTp.cells[:], technique=SWEEP)
    oTp.setElementType(elemTypes=element_type,regions=oTp.sets['entire'])
    oTp.seedPart(deviationFactor = 0.1, minSizeFactor = 0.1, size = oTpMesh)
    oTp.generateMesh()

    # use TET mesh in the timberPart in the dowel-zone
    if dowelModelling in ('1Ring','2Ring'):
        #Create partitions on part to achieve smooth mesh by diving the outer timber part into four parts
        tpPlaneXZ = tp.DatumPlaneByThreePoints(((bcx+a2/2,bcy+a1/2,0.0)),((bcx+a2/2+d/2,bcy+a1/2,0.0)),((bcx+a2/2+d/2,bcy+a1/2,b)))
        tpPlaneXY = tp.DatumPlaneByThreePoints(((bcx+a2/2,bcy+a1/2,0.0)),((bcx+a2/2,bcy+a1/2+d/2,0.0)),((bcx+a2/2,bcy+a1/2+d/2,b)))
        tp.PartitionCellByDatumPlane(cells=tp.cells[:],datumPlane = tp.datums[tpPlaneXZ.id])
        tp.PartitionCellByDatumPlane(cells=tp.cells[:],datumPlane = tp.datums[tpPlaneXY.id])
    tp.setMeshControls(elemShape=TET, regions=tp.cells[:], technique=FREE)
    tp.setElementType(elemTypes=element_type, regions=tp.sets['entire'])
    tp.seedPart(deviationFactor = 0.1, minSizeFactor = 0.1, size = tpMesh)
    tp.generateMesh()

    dowel.setMeshControls(elemShape=HEX_DOMINATED, regions=dowel.cells[:], technique=SWEEP)
    dowel.setElementType(elemTypes=element_type, regions=dowel.sets['entire'])
    dowel.seedPart(deviationFactor = 0.1, minSizeFactor = 0.1, size = dowelMesh)
    dowel.generateMesh()

    steelPl.setMeshControls(elemShape=HEX_DOMINATED, regions=steelPl.cells[:], technique=STRUCTURED)
    steelPl.setElementType(elemTypes=element_type, regions=steelPl.sets['entire'])
    steelPl.seedPart(deviationFactor = 0.1, minSizeFactor = 0.1, size = stPlMesh)
    steelPl.generateMesh()

    if dowelModelling in ('1Ring','2Ring'):
        ring1.setMeshControls(elemShape=HEX_DOMINATED, regions=ring1.cells[:], technique=SWEEP)
        ring1.setElementType(elemTypes=element_type, regions=ring1.sets['entire'])
        ring1.seedPart(deviationFactor = 0.1, minSizeFactor = 0.1, size = ring1Mesh)
        ring1.generateMesh()
    if dowelModelling == '2Ring':
        #Create partitions on part to achieve smooth mesh by diving the outer-ring into four parts
        r2PlaneXZ = ring2.DatumPlaneByThreePoints(((bcx+a2/2,bcy+a1/2,0.0)),((bcx+a2/2+d/2,bcy+a1/2,0.0)),((bcx+a2/2+d/2,bcy+a1/2,b)))
        r2PlaneXY = ring2.DatumPlaneByThreePoints(((bcx+a2/2,bcy+a1/2,0.0)),((bcx+a2/2,bcy+a1/2+d/2,0.0)),((bcx+a2/2,bcy+a1/2+d/2,b)))
        ring2.PartitionCellByDatumPlane(cells=ring2.cells[:],datumPlane = ring2.datums[r2PlaneXZ.id])
        ring2.PartitionCellByDatumPlane(cells=ring2.cells[:],datumPlane = ring2.datums[r2PlaneXY.id])
        ring2.setMeshControls(elemShape=HEX_DOMINATED, regions=ring2.cells[:], technique=SWEEP)
        ring2.setElementType(elemTypes=element_type,regions=ring2.sets['entire'])
        ring2.seedPart(deviationFactor = 0.1, minSizeFactor = 0.1, size = ring2Mesh)
        ring2.generateMesh()

    # based on the chosen input the correct output of the function is prepared
    if dowelModelling == '0Ring':
        parts = oTp, tp, dowel, steelPl
    elif dowelModelling == '1Ring':
        parts = oTp, tp, dowel, steelPl, ring1
    elif dowelModelling == '2Ring':
        parts = oTp, tp, dowel, steelPl, ring1, ring2

    return parts


def make_sections(model, parts, dowelMat, ringMat, tMat, timberSetup, dowelModelling):
    """The section function takes in the parts that are generated in the make-geometry function.
    It also takes in the materials that are to be assigned to the sections as well as the setup
    that is tested, and the chosen modelling of the dowel-zone.

    Args:
```

```
        model (_type_): the abaqus model that is built
        parts (_type_): the output of the make_geometry function
        dowelMat (_type_): material parameters of the dowel and the steel-plates
        ringMat (_type_): materialparameters of the rings
        tMat (_type_): the timber materialparameters
        timberSetup (_type_): setup corresponding either to lab-test: S1, S2, S3. Or Dorn or manual.
        dowelModelling (_type_): 'oring', '1ring' or '2ring'
    """

    oTp, tp, dowel, steelPl = parts[0],parts[1],parts[2],parts[3]
    if dowelModelling == '1Ring':
        ring1 = parts[4]
    elif dowelModelling == '2Ring':
        ring1 = parts[4]
        ring2 = parts[5]
    # create material, create and assign sketchOptions
    E_steel, nu_steel, fu_steel, fu_pl_steel = dowelMat[0], dowelMat[1], dowelMat[2], dowelMat[3]
    E_r1, nu_r1, adj_E_r2, adj_G_r2   =  ringMat[0], ringMat[1], ringMat[2], ringMat[3]
    E11,E22,E33,nu23,nu13,nu12,G23,G13,G12   =  tMat[0], tMat[1], tMat[2], tMat[3], tMat[4], tMat[5], tMat[6], tMat[7], tMat[8]

    # For the timber part
    mat = model.Material(name='wood')
    #mat.Elastic(table=((10000.0, 800.0, 400.0, 0.5, 0.6, 0.6, 600.0, 600.0, 30.0), ), type=ENGINEERING_CONSTANTS)
    mat.Elastic(table=((E11, E22, E33, nu12, nu13, nu23, G12, G13, G23), ), type=ENGINEERING_CONSTANTS)

    if timberSetup == 'S3':
        timberAngle = 0
    else:
        timberAngle = 90

    model.HomogeneousSolidSection(name='wood', material='wood', thickness=None)
    tp.SectionAssignment(region=tp.sets['entire'], sectionName='wood',
                         thicknessAssignment=FROM_SECTION)
    tp.MaterialOrientation(region = tp.sets['entire'], localCsys=None, axis = AXIS_3, angle= timberAngle )

    oTp.SectionAssignment(region=oTp.sets['entire'], sectionName='wood',
                         thicknessAssignment=FROM_SECTION)
    oTp.MaterialOrientation(region = oTp.sets['entire'], localCsys=None, axis = AXIS_3, angle= timberAngle )

    # For the dowel
    mat2 = model.Material(name='steel')
    mat2.Elastic(table=((E_steel, nu_steel), ))
    mat2.Plastic(table=((fu_steel, 0.0), ))

    model.HomogeneousSolidSection(name='steel', material='steel', thickness=None)
    dowel.SectionAssignment(region=dowel.sets['entire'], sectionName='steel', thicknessAssignment=FROM_SECTION)

    # For the steel plate
    mat3 = model.Material(name='steelPlate')
    mat3.Elastic(table=((E_steel, nu_steel), ))
    mat3.Plastic(table=((fu_pl_steel, 0.0), ))

    model.HomogeneousSolidSection(name='steelPlate', material='steelPlate', thickness=None)
    steelPl.SectionAssignment(region=steelPl.sets['entire'], sectionName='steelPlate', thicknessAssignment=FROM_SECTION)

    # if rings are modelled add correct materials
    if dowelModelling in ('1Ring','2Ring'):
        # For the inner ring
        mat4 = model.Material(name='ring1')
        mat4.Elastic(table=((E_r1, nu_r1), ))


        model.HomogeneousSolidSection(name='ring1', material='ring1', thickness=None)
        ring1.SectionAssignment(region=ring1.sets['entire'], sectionName='ring1', thicknessAssignment=FROM_SECTION)

    if dowelModelling == '2Ring':
        # For the outer ring
        mat5 = model.Material(name='ring2')
        mat5.Elastic(table=((adj_E_r2*E11, adj_E_r2*E22, adj_E_r2*E33, nu12, nu13, nu23, adj_G_r2*G12, adj_G_r2*G13, adj_G_r2*G23), ),
type=ENGINEERING_CONSTANTS)

        ring2.MaterialOrientation(region = ring2.sets['entire'], localCsys=None, axis = AXIS_3, angle= timberAngle )
        model.HomogeneousSolidSection(name='ring2', material='ring2', thickness=None)
        ring2.SectionAssignment(region=ring2.sets['entire'], sectionName='ring2', thicknessAssignment=FROM_SECTION)
    return

def make_assembly(model,parts,timberGeom, plateGeom, timberSetup, dowelModelling):
    """Generate the assembly from the parts that are created in the previous functions. Depending on the number of dowels
    a linear instance pattern is made that corresponds with the timber setup.

    Args:
        model (object): the abaqus model that is built
        parts (list): the output of the make_geometry function
        timberGeom (list of float): the geometry of the timber-part
        plateGeom (list of float): the geometry of the steel-plate
        timberSetup (string): setup corresponding either to lab-test: S1, S2, S3. Or Dorn or manual.
        dowelModelling (string): 'oring', '1ring' or '2ring'

```

```python
    Returns:
        aParts: assembly of the parts based on given dowelModelling
    """


    oTp, tp, dowel, steelPl = parts[0],parts[1],parts[2],parts[3]
    if dowelModelling in ('1Ring','2Ring'):
        ring1 = parts[4]
    if dowelModelling == '2Ring':
        ring2 = parts[5]
     # # Unpack the timber-geometry
    b,h,a1,a2,n_row,n_col = timberGeom[0], timberGeom[1],timberGeom[4],timberGeom[5],timberGeom[10],timberGeom[11]
    plLOut,plW,plGap      = plateGeom[2], plateGeom[3], plateGeom[8]

    # Define the assembly
    a = model.rootAssembly
    outerTimberPart = a.Instance(name = 'outerTimberPart', part = oTp    , dependent = ON)
    timberPart      = a.Instance(name = 'timberPart'     , part = tp     , dependent = ON)
    dowels          = a.Instance(name = 'dowel'          , part = dowel  , dependent = ON)
    if dowelModelling in ('1Ring','2Ring'):
        innerRing   = a.Instance(name ='innerRing'       , part = ring1  , dependent = ON)
    if dowelModelling == '2Ring':
        outerRing   = a.Instance(name ='outerRing'       , part = ring2  , dependent = ON)
    steelPlate      = a.Instance(name ='steelPlate'      , part = steelPl, dependent = ON)

    # Translate the steel plate to correct coordinates
    a.translate(instanceList=('steelPlate', ), vector=((h-plW)/2, 0.0, (b+plGap)))

    # Insert dowel regions into their locations
    if dowelModelling == '0Ring':
        listOfInstances = ['timberPart','dowel']
    if dowelModelling == '1Ring':
        listOfInstances = ['timberPart','dowel','innerRing']
    if dowelModelling == '2Ring':
        listOfInstances = ['timberPart','dowel','innerRing','outerRing']
    dowelstest      = a.LinearInstancePattern(instanceList=listOfInstances,number1=n_col,spacing1=a2,number2=n_row,spacing2=a1)
    # turn off the datum-planes to clean up the view of the model
    session.viewports['Viewport: 1'].assemblyDisplay.geometryOptions.setValues(datumPlanes=OFF)

    model.Coupling(controlPoint=steelPlate.sets['SteelPL_Bot'], couplingType=KINEMATIC,
        influenceRadius=WHOLE_SURFACE, localCsys=None, name='RP_coupling', surface=steelPlate.surfaces['plateEnd2']
        , u1=ON, u2=ON, u3=ON, ur1=ON, ur2=ON, ur3=ON)
    if timberSetup == 'S3':
        model.Coupling(controlPoint=outerTimberPart.sets['S3-disp'], couplingType=KINEMATIC,
        influenceRadius=WHOLE_SURFACE, localCsys=None, name='RP_coupling-OTP', surface=outerTimberPart.surfaces['S3-disp']
        , u1=ON, u2=ON, u3=ON, ur1=ON, ur2=ON, ur3=ON)

    if dowelModelling == '0Ring':
        aParts = outerTimberPart, timberPart, dowels,  steelPlate
    elif dowelModelling == '1Ring':
        aParts = outerTimberPart, timberPart, dowels,  steelPlate, innerRing
    elif dowelModelling == '2Ring':
        aParts = outerTimberPart, timberPart, dowels,  steelPlate, innerRing, outerRing

    return aParts


def make_boundaries(model, aParts, fy, TOL, timberGeom,plateGeom, timberSetup, dowelModelling, analysisType):
    """In the make_boundaries function the boundary conditions are specified: symmetry conditions, as well as
    where the model is fixed, and where to apply the loading. We also make the step and the history output
    requests. The last part of the function ties all the different parts in the model together using modelling
    ties and a lot of conditional statements to tie the correct parts together.

    Args:
        model (object): _description_
        aParts (list): the assembled parts
        fy (float): force  in y-direction
        TOL (float): tolerance
        timberGeom (list of float): the geometry of the timber-part
        plateGeom (list of float): the geometry of the steel-plate
        timberSetup (string): setup corresponding either to lab-test: S1, S2, S3. Or Dorn or manual.
        dowelModelling (string): 'oring', '1ring' or '2ring'
        analysisType (string): Wether to use a force or a prescribed displacement as loading
    """

    outerTimberPart, timberPart, dowels,  steelPlate = aParts[0],aParts[1],aParts[2],aParts[3],

    if dowelModelling == '2Ring':
        outerRing = aParts[5]
    if dowelModelling in ('1Ring','2Ring'):
        innerRing = aParts[4]
        partToConnect = innerRing
    else:
        partToConnect = timberPart

    n_row, n_col, l  = timberGeom[10], timberGeom[11], timberGeom[2]
    plLOut = plateGeom[2]

    # create step
```

```python
    if analysisType == 'ForceDisp':
        step = model.StaticStep(name='forceApp', previous='Initial', maxNumInc=1000,
                                initialInc = 0.05, minInc=1e-08, maxInc =0.05, nlgeom=ON)
    else:
        step = model.StaticStep(name='forceApp', previous='Initial', maxNumInc=1000,
                                initialInc = 0.1, minInc=1e-08, maxInc =0.1, nlgeom=ON)

    # create different symmetry BC, load and history output requests based on the timberSetup
    if timberSetup == 'S3':
        model.XsymmBC(createStepName='Initial', localCsys=None, name='Timber fastening', region=outerTimberPart.sets['S3fix'])

        if analysisType == 'ForceDisp':
            model.DisplacementBC(amplitude=UNSET, createStepName='forceApp'
            , distributionType=UNIFORM, fieldName='', fixed=OFF, localCsys=None, name=
            'Displacement-load', region=steelPlate.sets['SteelPL_Bot'], u1=UNSET, u2=-6.0, u3=UNSET, ur1=UNSET, ur2=UNSET, ur3=UNSET)
            model.HistoryOutputRequest(name='ForceDisp', createStepName = 'forceApp', variables = ('RF2','U2',), region=steelPlate.sets['SteelPL_Bot'])
            model.HistoryOutputRequest(name='S3-Disp', createStepName = 'forceApp', variables = ('U2',), region=outerTimberPart.sets['S3-disp'])
        else:
            model.Pressure(amplitude=UNSET, createStepName='forceApp',distributionType=TOTAL_FORCE, field='', magnitude=-fy, name='Load-1', region = steelPlate.surfaces['plateEnd2'])
            model.HistoryOutputRequest(name='DispEnd', createStepName = 'forceApp', variables = ('U2',), region=steelPlate.sets['SteelPL_Bot'])
            model.HistoryOutputRequest(name='S3-Disp', createStepName = 'forceApp', variables = ('U2',), region=outerTimberPart.sets['S3-disp'])
        for i in range(2): model.DisplacementBC(amplitude=UNSET, createStepName='forceApp', distributionType=UNIFORM, fieldName='', fixed=OFF, localCsys=None, name='holdDown'+str(i+1), region=outerTimberPart.sets['holdDown'+str(i+1)], u1=UNSET, u2=0.0, u3=UNSET, ur1=UNSET, ur2=UNSET, ur3=UNSET)
        # Create request for output of the U2 displacement of the top surface


    else:
        # Define boundaries and load
        topSurf = model.rootAssembly.Surface(name='topSurf', side1Faces = outerTimberPart.faces.getByBoundingBox(yMin = float(l)-TOL))
        if analysisType == 'ForceDisp':
            model.DisplacementBC(amplitude=UNSET, createStepName='forceApp'
            , distributionType=UNIFORM, fieldName='', fixed=OFF, localCsys=None, name=
            'Displacement-load', region=outerTimberPart.sets['top'], u1=UNSET, u2=2.0, u3=UNSET, ur1=UNSET, ur2=UNSET, ur3=UNSET)
            model.HistoryOutputRequest(createStepName='forceApp', name='Force', rebar=EXCLUDE, region=steelPlate.sets['SteelPL_Bot'], sectionPoints=DEFAULT, variables=('RF2', ))
        else:
            model.Pressure(amplitude=UNSET, createStepName='forceApp',distributionType=TOTAL_FORCE, field='', magnitude=-fy, name='Load-1', region=topSurf)
            model.HistoryOutputRequest(name='DispTopPlate', createStepName = 'forceApp', variables = ('U2',), region=steelPlate.sets['plateTop2'])
        model.EncastreBC(createStepName='Initial', localCsys=None, name='PlateFixed', region= steelPlate.sets['SteelPL_Bot'])
        # Create request for output of the U2 displacement of the top surface
        model.HistoryOutputRequest(name='DispTop', createStepName = 'forceApp', variables = ('U2',), region=outerTimberPart.sets['top'])

    # Fix the edge of the plate


    model.ZsymmBC(createStepName='Initial', localCsys=None, name='Sym-steelplate'     , region=steelPlate.sets['symPlane'])
    model.ZsymmBC(createStepName='Initial', localCsys=None, name='Sym-timber-midplane', region=outerTimberPart.sets['sym-midplane-BC'])
    # Add the BC on each dowel and tie together the different circular parts of the "building blocks"
    for i in range(1,n_col+1,1):
        for j in range(1,n_row+1,1):
            if i==1 and j==1:
                model.ZsymmBC(createStepName='Initial', localCsys=None, name='DowelHoldSym-'+str(i)+'-'+str(j), region=dowels.sets['dowelEnd'])
                model.Tie(master=dowels.surfaces[     'dowelSurf'    ], name='D-R1 Tie-'+str(i)+'-'+str(j)    , slave=partToConnect.surfaces[  'innerRingIN'  ])
                model.Tie(master=dowels.surfaces[     'dowelSurfPl' ], name='D-Pl Tie-'+str(i)+'-'+str(j)    , slave=steelPlate.surfaces['dowelTiePl_'+str(j)+'-'+str(i)])
                if dowelModelling == '1Ring':
                    model.Tie(master=partToConnect.surfaces[ 'innerRingOUT'], name='R1-T Tie-'+str(i)+'-'+str(j)   , slave=timberPart.surfaces[  'timberIN'   ])
                if dowelModelling == '2Ring':
                    model.Tie(master=innerRing.surfaces[ 'innerRingOUT'], name='R1-R2 Tie-'+str(i)+'-'+str(j)   , slave=outerRing.surfaces[  'outerRingIN'  ])
                    model.Tie(master=outerRing.surfaces[ 'outerRingOUT'], name='R2-T1 Tie-'+str(i)+'-'+str(j)   , slave=timberPart.surfaces[ 'timberIN'     ])
            else:
                model.ZsymmBC(createStepName='Initial', localCsys=None, name='DowelHoldSym-'+str(i)+'-'+str(j), region=model.rootAssembly.instances['dowel-lin-'+str(i)+'-'+str(j)].sets['dowelEnd'])
                model.Tie(master=model.rootAssembly.instances['dowel-lin-'+str(i)+'-'+str(j)].surfaces['dowelSurfPl']      , name = 'D-Pl Tie-'+str(i)+'-'+str(j)    , slave=steelPlate.surfaces[   'dowelTiePl_'+str(j)+'-'+str(i)  ])
                if dowelModelling == '0Ring':
                    model.Tie(master=model.rootAssembly.instances['dowel-lin-'+str(i)+'-'+str(j)].surfaces['dowelSurf'], name='D-T Tie-'+str(i)+'-'+str(j)   , slave=model.rootAssembly.instances['timberPart-lin-'+str(i)+'-'+str(j)].surfaces['innerRingIN'])
                if dowelModelling == '1Ring':
                    model.Tie(master=model.rootAssembly.instances['innerRing-lin-'+str(i)+'-'+str(j)].surfaces['innerRingOUT'], name='R1-T Tie-'+str(i)+'-'+str(j)   , slave=model.rootAssembly.instances['timberPart-lin-'+str(i)+'-'+str(j)].surfaces['timberIN'])
                if dowelModelling in ('1Ring','2Ring'):
                    model.Tie(master=model.rootAssembly.instances['dowel-lin-'+str(i)+'-'+str(j)].surfaces['dowelSurf']        , name = 'D-R1 Tie-'+str(i)+'-'+str(j)    , slave=model.rootAssembly.instances['innerRing-lin-'+str(i)+'-'+str(j)].surfaces['innerRingIN'])
                if dowelModelling == '2Ring':

                    model.Tie(master=model.rootAssembly.instances['innerRing-lin-'+str(i)+'-'+str(j)].surfaces['innerRingOUT'], name = 'R1-R2 Tie-'+str(i)+'-'+str(j)   , slave=model.rootAssembly.instances['outerRing-lin-'+str(i)+'-'+str(j)].surfaces['outerRingIN'])
```

```python
                    model.Tie(master=model.rootAssembly.instances['outerRing-lin-'+str(i)+'-'+str(j)].surfaces['outerRingOUT'], name = 'R2-T1 Tie-'+str(i)+'-'+str(j)  , slave=model.rootAssembly.instances['timberPart-lin-'+str(i)+'-'+str(j)].surfaces['timberIN'])

    for i in range(1,n_col+1,1):
        for j in range(1,n_row+1,1):
            if i==1 and j==1:           # Check if i am in the first basis-block
                model.Tie(master=model.rootAssembly.instances['timberPart'].surfaces['timberOUT_B'], name = 'T1-T2_B Tie-'+str(i)+'-'+str(j)    ,  slave=model.rootAssembly.instances['outerTimberPart'].surfaces['outerTimberIN-B'+str(i)])
                model.Tie(master=model.rootAssembly.instances['timberPart'].surfaces['timberOUT_L'], name = 'T1-T2_L Tie-'+str(i)+'-'+str(j)    ,  slave=model.rootAssembly.instances['outerTimberPart'].surfaces['outerTimberIN-L'+str(i)])
                if n_col==1:            # If there is only one column R tie connects directly to the outerTimberPart
                    model.Tie(master=model.rootAssembly.instances['timberPart'].surfaces['timberOUT_R'], name = 'T1-T2_R Tie-'+str(i)+'-'+str(j),  slave=model.rootAssembly.instances['outerTimberPart'].surfaces['outerTimberIN-R'+str(i)])
                else:                   # Else we adjust R tie to connect to the block on the side of it
                    model.Tie(master=model.rootAssembly.instances['timberPart'].surfaces['timberOUT_R'], name = 'T1-T1_IR Tie-'+str(i)+'-'+str(j), slave=model.rootAssembly.instances['timberPart-lin-'+str(i+1)+'-'+str(j)].surfaces['timberOUT_L'])
                if n_row==1:            # If we only have one row adjust top connection to connect to outerTimberPart
                    model.Tie(master=model.rootAssembly.instances['timberPart'].surfaces['timberOUT_T'], name = 'T1-T2_T Tie-'+str(i)+'-'+str(j) , slave=model.rootAssembly.instances['outerTimberPart'].surfaces['outerTimberIN-T'+str(i)])
                else:                   # Else we connect the top to the block over it
                    model.Tie(master=model.rootAssembly.instances['timberPart'].surfaces['timberOUT_T'], name = 'T1-T1_IT Tie-'+str(i)+'-'+str(j), slave=model.rootAssembly.instances['timberPart-lin-'+str(i)+'-'+str(j+1)].surfaces['timberOUT_B'])
            else:                       # Else we are in the regularly numbered blocks
                if j==1:                # For the other blocks at the bottom
                    model.Tie(master=model.rootAssembly.instances['timberPart-lin-'+str(i)+'-'+str(j)].surfaces['timberOUT_B'], name = 'T1-T2_B Tie-'+str(i)+'-'+str(j), slave=model.rootAssembly.instances['outerTimberPart'].surfaces['outerTimberIN-B'+str(i)])
                if i==1:                # Adjust to connect L to the outerTimberPart in first column
                    model.Tie(master=model.rootAssembly.instances['timberPart-lin-'+str(i)+'-'+str(j)].surfaces['timberOUT_L'], name = 'T1-T2_L Tie-'+str(i)+'-'+str(j), slave=model.rootAssembly.instances['outerTimberPart'].surfaces['outerTimberIN-L'+str(j)])
                if i==n_col:            # Adjust the connection to the right if we are in the last column
                    model.Tie(master=model.rootAssembly.instances['timberPart-lin-'+str(i)+'-'+str(j)].surfaces['timberOUT_R'], name = 'T1-T2_R Tie-'+str(i)+'-'+str(j), slave=model.rootAssembly.instances['outerTimberPart'].surfaces['outerTimberIN-R'+str(j)])
                else:                   # Else connect to the block to the right of it
                    model.Tie(master=model.rootAssembly.instances['timberPart-lin-'+str(i)+'-'+str(j)].surfaces['timberOUT_R'], name = 'T1-T1_IR Tie-'+str(i)+'-'+str(j),slave=model.rootAssembly.instances['timberPart-lin-'+str(i+1)+'-'+str(j)].surfaces['timberOUT_L'])
                if n_row==j:            # Adjust if we are on top row to connect to outer timberPart
                    model.Tie(master=model.rootAssembly.instances['timberPart-lin-'+str(i)+'-'+str(j)].surfaces['timberOUT_T'], name = 'T1-T2_T Tie-'+str(i)+'-'+str(j), slave=model.rootAssembly.instances['outerTimberPart'].surfaces['outerTimberIN-T'+str(i)])
                else:                   # Else we connect to the block over it
                    model.Tie(master=model.rootAssembly.instances['timberPart-lin-'+str(i)+'-'+str(j)].surfaces['timberOUT_T'], name = 'T1-T1_IT Tie-'+str(i)+'-'+str(j),slave=model.rootAssembly.instances['timberPart-lin-'+str(i)+'-'+str(j+1)].surfaces['timberOUT_B'])

    return

def run_model(model1, job_name):
    """This function creates the job, with description of calculation settings,
    submit the job and then wait for the job to complete before the rest of the
    code can be executed.

    Args:
        model1 (object): the abaqus model that is created
        job_name (string): the nmame of the job
    """
    # job = mdb.Job(name = job_name, model = model1, type = ANALYSIS, resultsFormat=ODB)
    job = mdb.Job(atTime=None, contactPrint=OFF, description='', echoPrint=OFF,
    explicitPrecision=SINGLE, getMemoryFromAnalysis=True, historyPrint=OFF,
    memory=90, memoryUnits=PERCENTAGE, model=model1, modelPrint=OFF,
    multiprocessingMode=DEFAULT, name=job_name, nodalOutputPrecision=SINGLE,
    numCpus=3, numDomains=3, numGPUs=1, queue=None, resultsFormat=ODB, scratch=
    '', type=ANALYSIS, userSubroutine='', waitHours=0, waitMinutes=0)
    job.submit(consistencyChecking = OFF)
    job.waitForCompletion()
    return


def evaluate_results(job_name, dowelConfig, timberSetup):
    """The evaluate_results function generates png plots of stresses and
    displacement in the model, and saves them.


    Args:
        job_name (string): name of the job
        dowelConfig (list): first item number of rows, second number of columns
        timberSetup (string): setup corresponding either to lab-test: S1, S2, S3. Or Dorn or manual.
    """
    # Make sure the outpath exist, or create a folder to store the plots
    user = getpass.getuser()
    if not os.path.exists('C:\Users\\'+user+'\OneDrive - NTNU\Documents\Results abaqus\\Plots'):
            os.makedirs('C:\Users\\'+user+'\OneDrive - NTNU\Documents\Results abaqus\\Plots')
    pathTosave = 'C:\Users\\'+user+'\OneDrive - NTNU\Documents\Results abaqus\\Plots\\'

    n_dowels = dowelConfig[0]*dowelConfig[-1]

    # open the odb file
    odb = session.openOdb(job_name+'.odb')
    vp  = session.viewports['Viewport: 1']
    vp.setValues(displayedObject=odb)

    vp.restore()
    # position of the viwport and size
    vp.setValues(origin=(-20,0))
```

```
    vp.setValues(width=300, height=200)

    # set the right view
    vp.view.setValues(session.views['Right'])
    # view the right filed output
    vp.odbDisplay.setValues(viewCutNames=('X-Plane', 'X-Plane'), viewCut=ON)
    vp.odbDisplay.display.setValues(plotState=(CONTOURS_ON_DEF, ))
    vp.odbDisplay.setPrimaryVariable( variableLabel='S', outputPosition=INTEGRATION_POINT, refinement=(INVARIANT, 'Mises'), )
    if timberSetup == 'S3':
        vp.view.setValues(nearPlane=1000.00, farPlane=1700.00, width=300.00, height=245.00, viewOffsetX=50.000,viewOffsetY=-20.00)
    else:
        vp.view.setValues(nearPlane=1000.00, farPlane=1700.00, width=350.00, height=295.00, viewOffsetX=50.000,viewOffsetY=0.00)
    # change the legend and what is displayed
    vp.viewportAnnotationOptions.setValues(legendFont=
        '-*-verdana-medium-r-normal-*-*-140-*-*-p-*-*-*')
    vp.viewportAnnotationOptions.setValues(triad=OFF, state=OFF,
        legendBackgroundStyle=MATCH,annotations=OFF,compass=OFF,
        title=OFF)
    # print viewport to png file
    session.printOptions.setValues(reduceColors=False, vpDecorations=OFF)
    session.pngOptions.setValues(imageSize=(2000, 2500))
    session.printToFile(fileName=pathTosave+job_name+'_S_Mises_Xcut', format=PNG,
                canvasObjects=(vp,))
    odb.close()

    # make a cut-view of the stresses(new code)
    odb = session.openOdb(job_name+'.odb')
    vp  = session.viewports['Viewport: 1']
    vp.setValues(displayedObject=odb)
    # Change size of viewport (e.g. 300x200 pixel)
    vp.restore()
    # position of the viwport
    vp.setValues(origin=(-30,0))
    vp.setValues(width=300, height=200)

    if timberSetup == 'S3':
        vp.view.setValues(session.views['Front'])
    else:
        vp.view.setValues(session.views['Back'])
    # Make a cut in the part
    odbD = session.viewports['Viewport: 1'].odbDisplay
    odbD.display.setValues(plotState=(CONTOURS_ON_DEF, ))
    odbD.setPrimaryVariable( variableLabel='U', outputPosition=NODAL, refinement=(COMPONENT, 'U2'), )
    # print viewport to png file
    # change the legend and what is displayed
    vp.viewportAnnotationOptions.setValues(legendFont=
        '-*-verdana-medium-r-normal-*-*-140-*-*-p-*-*-*')
    vp.viewportAnnotationOptions.setValues(triad=OFF, state=OFF,
        legendBackgroundStyle=MATCH,annotations=OFF,compass=OFF,
        title=OFF)
    session.printOptions.setValues(reduceColors=False, vpDecorations=OFF)
    session.pngOptions.setValues(imageSize=(2000, 2500))
    session.printToFile(fileName=pathTosave+job_name+'_U2_cut_XY', format=PNG,canvasObjects=(vp,))
    odb.close()

    # S22 or S11 depending on timbersetup
    # make another cut-view of the stresses(new code)
    odb = session.openOdb(job_name+'.odb')
    vp  = session.viewports['Viewport: 1']
    vp.setValues(displayedObject=odb)
    # Change size of viewport (e.g. 300x200 pixel)
    vp.restore()
    # position of the viwport
    vp.setValues(origin=(-30,0))
    vp.setValues(width=300, height=200)
    vp.view.setValues(session.views['Front'])
    odbD.display.setValues(plotState=(CONTOURS_ON_DEF, ))

    # remove dowel, outer timber and steel plates to focus on the most utilized timberparts
    leaf = dgo.LeafFromPartInstance(partInstanceName=('DOWEL', ))
    session.viewports['Viewport: 1'].odbDisplay.displayGroup.remove(leaf=leaf)
    for i in range(dowelConfig[0]):
        for j in range(dowelConfig[1]):
            leaf = dgo.LeafFromPartInstance(partInstanceName=('DOWEL-LIN-'+str(j+1)+'-'+str(i+1), ))
            session.viewports['Viewport: 1'].odbDisplay.displayGroup.remove(leaf=leaf)
    leaf = dgo.LeafFromPartInstance(partInstanceName=('STEELPLATE', ))
    session.viewports['Viewport: 1'].odbDisplay.displayGroup.remove(leaf=leaf)
    # if only the inner part is wanted, uncomment the next two lines:
    # leaf = dgo.LeafFromPartInstance(partInstanceName=('OUTERTIMBERPART', ))
    # session.viewports['Viewport: 1'].odbDisplay.displayGroup.remove(leaf=leaf)

    if timberSetup == 'S3':
        odbD.setPrimaryVariable(variableLabel='S', outputPosition=INTEGRATION_POINT, refinement=(COMPONENT, 'S22'),)
    else:
        odbD.setPrimaryVariable(variableLabel='S', outputPosition=INTEGRATION_POINT, refinement=(COMPONENT, 'S11'),)
    # change the legend and what is displayed
    vp.viewportAnnotationOptions.setValues(legendFont=
        '-*-verdana-medium-r-normal-*-*-140-*-*-p-*-*-*')
    vp.viewportAnnotationOptions.setValues(triad=OFF, state=OFF,
```

```python
        legendBackgroundStyle=MATCH,annotations=OFF,compass=OFF,
        title=OFF)
    # print viewport to png file
    session.printOptions.setValues(reduceColors=False, vpDecorations=OFF)
    session.pngOptions.setValues(imageSize=(2000, 2500))
    session.printToFile(fileName=pathTosave+job_name+'_Stresses', format=PNG,canvasObjects=(vp,))
    odb.close()
    return



def evaluate_historyOutput(job_name, forcey, ringMat, opening_angle, configuration, timberSetup, analysisType, dowelModelling):
    """This function opens the odb from the analysis and takes out the output from the
    historyoutput requests made in the make_boundaries function. These datas are then
    either stored or plotted directly.

    Args:
        job_name (string): Auto generated to describe the given setup and analysis type
        forcey (int): The applied force in y-direction
        ringMat (int): The material in the modelling ring
        opening_angle (float): Contact angle between wood and dowel
        configuration (list): first item number of rows, second number of columns
        timberSetup (string): setup corresponding either to lab-test: S1, S2, S3. Or Dorn or manual.
        analysisType (string): Wether to use a force or a prescribed displacement as loading
        dowelModelling (string): 'oring', '1ring' or '2ring'

    Returns:
        List: List of prescribed outputs depending on analysis type.
    """


    odb = session.openOdb(job_name+'.odb')
    vp = session.viewports['Viewport: 1']
    vp.setValues(displayedObject=odb)
    # evaluate the history output
    # -----------------------------------------------------
    step = odb.steps['forceApp']
    n_row = configuration[0]
    n_col = configuration[1]
    n_dowels = n_row*n_col
    force = (float(forcey)/1000.0)

    if analysisType=='ForceDisp':
        rpForce = []
        rpDisp = []
        # Take out the forces from the steel-plate edge
        histReg_point = [i for i in step.historyRegions.values() if i.name.startswith('Node STEELPLATE')]
        for hr in histReg_point:
            res  = np.array(hr.historyOutputs['RF2'].data)
            rpForce.append(res[:,:][:,:])

        Force = [(-1)*(10**(-3))*forcei[:,-1] for forcei in rpForce]

        # Take out the displacement at the top of the outer-timberpart where the LVDT is fastened

        # the displacement that is specified as loading is then set
        if timberSetup == 'S3':
            histReg_point2 = [i for i in step.historyRegions.values() if i.name.startswith('Node OUTERTIMBERPART')]
            for hr in histReg_point2:
                res  = np.array(hr.historyOutputs['U2'].data)
                rpDisp = res[:,-1]

            disp = np.arange(0,6.3,0.3)#(0,2.1,0.1)
            # can then calculate the displacement, as the LVDT would measure it in the tests
            relDisp = disp+rpDisp

        else:
            disp = np.arange(0,2.1,0.1)
            relDisp = disp

        # The path must be updated to a path on your computer, if no path exists a new folder will be created
        user = getpass.getuser()
        if not os.path.exists('C:\Users\\'+user+'\OneDrive - NTNU\Documents\Results abaqus\\Text'):
            os.makedirs('C:\Users\\'+user+'\OneDrive - NTNU\Documents\Results abaqus\\Text')
            pathTosave = 'C:\Users\\'+user+'\OneDrive - NTNU\Documents\Results abaqus\\Text\\'
            pathToPrint = 'C:\Users\\'+user+'\OneDrive - NTNU\Documents\Master\Results abaqus\\'+str(timberSetup)+'-r-'+str(configuration[0])+'-c-'+str(configuration[1])+'-'+str(dowelModelling)+'_forceDisp.png'
        pathTosave = 'C:\Users\\'+user+'\OneDrive - NTNU\Documents\Master\Results abaqus\\Text\\'
        pathToPrint = 'C:\Users\\'+user+'\OneDrive - NTNU\Documents\Master\Results abaqus\\'+str(timberSetup)+'-r-'+str(configuration[0])+'-c-'+str(configuration[1])+'-'+str(dowelModelling)+'_forceDisp.png'

        np.savetxt(pathTosave+'Force '+str(timberSetup)+'-r-'+str(configuration[0])+'-c-'+str(configuration[1])+'-'+str(dowelModelling),Force)
        np.savetxt(pathTosave+'Disp '+str(timberSetup)+'-r-'+str(configuration[0])+'-c-'+str(configuration[1])+'-'+str(dowelModelling),relDisp)

        #  Plotting the force displacement data
        fig1, ax = plt.subplots(1,1 ,dpi=1200)
        ax.plot(relDisp,Force[-1],label = str(job_name))
```

```python
        ax.grid(zorder=1)
        ax.set_xlabel('Displacement [mm]')
        ax.set_ylabel('Force [kN]')
        ax.legend( fontsize=8) #loc = 'upper left',
        fig1.tight_layout()
        fig1.savefig(pathToPrint)
        plt.close(fig1)
    else:
        if timberSetup == 'S3':
            # do this
            histReg_point2 = [i for i in step.historyRegions.values() if i.name.startswith('Node OUTERTIMBERPART')]
            topWoodDisp, steelDisp = [],[]
            for hr in histReg_point2:
                res  = np.array(hr.historyOutputs['U2'].data)
                topWoodDisp.append(res[:,-1])
            histReg_point = [i for i in step.historyRegions.values() if i.name.startswith('Node STEELPLATE')]
            for hr in histReg_point:
                res  = np.array(hr.historyOutputs['U2'].data)
                steelDisp.append(res[:,:][:,-1])
            displacement = abs(steelDisp[0][-1]-topWoodDisp[0][-1])
            K_connection = force/displacement      # [kN/mm]
            K_perDowel = K_connection/n_dowels
            res_i = np.empty([1, 10])
            res_i = np.array([ n_row, n_col, n_dowels, K_perDowel, K_connection, displacement, force, opening_angle, ringMat[0], ringMat[2] ])
            print(res_i)
        else:

            # make a list of all the nodal history region objects by selecting the history region that starts with 'Node '
            hist_reg = [i for i in step.historyRegions.values() if i.name.startswith('Node OUTERTIMBERPART')]
            # then run the commands to extract final displacement in each node and calculate average disp in U2 direction
            nodalDisp = []
            for hr in hist_reg:
                res_u = np.array(hr.historyOutputs['U2'].data)
                nodalDisp.append(res_u[:,1][-1])

            topSurfDisp = np.average(nodalDisp)

            # make a list of all the nodal history region objects by selecting the history region that starts with 'Node '
            hist_reg2 = [i for i in step.historyRegions.values() if i.name.startswith('Node STEELPLATE')]
            # then run the commands to extract final displacement in each node and calculate average disp in U2 direction
            nodalDisp2 = []
            for hr in hist_reg2:
                res_u2 = np.array(hr.historyOutputs['U2'].data)
                nodalDisp2.append(res_u2[:,1][-1])

            STEELtopSurfDisp = np.average(nodalDisp2)
            relativeDisp = topSurfDisp-STEELtopSurfDisp
            # calculate the stiffness in the end



            K_connection = force/topSurfDisp      # [kN/mm]
            K_relative = force/relativeDisp
            print('Displacement of topTimber and steelplate:')
            print(topSurfDisp,STEELtopSurfDisp)
            print('Relative displacement(as measured in lab)')
            print(topSurfDisp-STEELtopSurfDisp)
            print('The relative stiffness is:')
            print(K_relative)
            print('The relative stiffness per dowel and shear plane is:')
            print(K_relative/n_dowels)
            print(K_connection, topSurfDisp)

            K_perDowel = K_connection/n_dowels
            # write the data to a array containing the following

            res_i = np.empty([1, 10])
            res_i = np.array([ n_row, n_col, n_dowels, K_perDowel, K_connection, topSurfDisp, force, opening_angle, ringMat[0], ringMat[2] ])
            print(res_i)

        assembly = odb.rootAssembly
        numNodes = numElements = 0
        for name, instance in assembly.instances.items():
            n = len(instance.nodes)
            print('Number of nodes of instance %s: %d' % (name, n))
            numNodes = numNodes + n
            n = len(instance.elements)
            print('Number of elements of instance %s: %d'%(name,n))
            numElements = numElements + n

        print('Number of instances     : %d'%(len(assembly.instances)))
        print('Total number of elements: %d'%(numElements))
        print('Total number of nodes   : %d'%(numNodes))


        modelProperties = np.array([numElements, numNodes, K_perDowel,None])

```

```python
    odb.close()
    if analysisType == 'ForceDisp':
        return Force
    else:
        return res_i, modelProperties


###########################################################################################
##                              Input parameters                                        ##
###########################################################################################

def input_parameters(specimen, config, opening_angle, meshS):
    """Function where all the input parameters such as length, width, material parameters and
    so on are stored and given as input in the other functions.

    Args:
        specimen (string): setup corresponding either to lab-test: S1, S2, S3. Or Dorn or manual.
        config (list of INT): first item number of rows, second number of columns
        opening_angle (float): Angle of the tie connection between dowel and innner timberpart

    Returns:
        lists: All geometries, materials and specified forces.
    """

    # Create a dictionary that stores the setup for S1,S2,S3, Dorn and manual setup
    tSetup = {
        'S1' : {
            'b'  : 51.0  ,                  # Width of the timber part  [mm]
            'h'  : 90.0  ,                  # Height of the timber part [mm]
            'l'  : 320.0 ,                  # length of the timber part [mm]
            'd'  : 12.0  ,                  # Diameter of the dowels    [mm]
            'a1' : 60.0  ,                  # Distance between dowels in grain dir         [mm]
            'a2' : 45.0  ,                  # Distance between dowels normal to grain dir  [mm]
            'e1' : 100.0 ,                  # End distance from dowel in grain dir         [mm]
            'e2' : 45.0  ,                  # Side distance from dowel normal to grain dir [mm]
            'n_row' : config[0],            # Number of rows of dowels    []
            'n_col' : 1                     # Number of colons of dowels []
        },
        'S2' : {
            'b'  : 51.0  ,                  # Width of the timber part  [mm]
            'h'  : 220.0 ,                  # Height of the timber part [mm]
            'l'  : 320.0 ,                  # length of the timber part [mm]
            'd'  : 12.0  ,                  # Diameter of the dowels    [mm]
            'a1' : 60.0  ,                  # Distance between dowels in grain dir         [mm]
            'a2' : 55.0  ,                  # Distance between dowels normal to grain dir  [mm]
            'e1' : 100.0 ,                  # End distance from dowel in grain dir         [mm]
            'e2' : 55.0  ,                  # Side distance from dowel normal to grain dir [mm]
            'n_row' : config[0],            # Number of rows of dowels    []
            'n_col' : config[1]             # Number of colons of dowels []
        },
        'S3' : {
            'b'  : 51.0  ,                  # Width of the timber part  [mm]
            'h'  : 600.0 ,                  # Height of the timber part [mm]
            'l'  : 300.0 ,                  # length of the timber part [mm]
            'd'  : 12.0  ,                  # Diameter of the dowels    [mm]
            'a1' : 55.0  ,                  # Distance between dowels in grain dir         [mm]
            'a2' : 60.0  ,                  # Distance between dowels normal to grain dir  [mm]
            'e1' : 95.0 ,                   # End distance from dowel in grain dir         [mm]
            'e2' : 220.0  ,                 # Side distance from dowel normal to grain dir [mm]
            'n_row' : config[0],            # Number of rows of dowels    []
            'n_col' : config[1]             # Number of colons of dowels []
        },
        'Dorn' : {
            'b'  : 45.0  ,                  # Width of the timber part  [mm]
            'h'  : 72.0 ,                   # Height of the timber part [mm]
            'l'  : 400.0 ,                  # length of the timber part [mm]
            'd'  : 12.0  ,                  # Diameter of the dowels    [mm]
            'a1' : 60.0  ,                  # Distance between dowels in grain dir         [mm]
            'a2' : 55.0  ,                  # Distance between dowels normal to grain dir  [mm]
            'e1' : 84.0  ,                  # End distance from dowel in grain dir         [mm]
            'e2' : 36.0  ,                  # Side distance from dowel normal to grain dir [mm]
            'n_row' : 1  ,                  # Number of rows of dowels    []
            'n_col' : 1                     # Number of colons of dowels []
        },
        'Manual' : {
            'b'  : 51.0  ,                  # Width of the timber part  [mm]
            'h'  : 90.0 ,                   # Height of the timber part [mm]
            'l'  : 800.0 ,                  # length of the timber part [mm]
            'd'  : 12.0  ,                  # Diameter of the dowels    [mm]
            'a1' : 60.0  ,                  # Distance between dowels in grain dir         [mm]
            'a2' : 45.0  ,                  # Distance between dowels normal to grain dir  [mm]
            'e1' : 100.0 ,                  # End distance from dowel in grain dir         [mm]
            'e2' : 45.0  ,                  # Side distance from dowel normal to grain dir [mm]
            'n_row' : config[0],            # Number of rows of dowels    []
            'n_col' : config[1]             # Number of colons of dowels []
        }
    }

    b  = tSetup[specimen]['b']              # Width of the timber part  [mm]
```

```python
    h  = tSetup[specimen]['h']                # Height of the timber part [mm]
    l  = tSetup[specimen]['l']                # length of the timber part [mm]
    d  = tSetup[specimen]['d']                # Diameter of the dowels    [mm]
    a1 = tSetup[specimen]['a1']               # Distance between dowels in grain dir         [mm]
    a2 = tSetup[specimen]['a2']               # Distance between dowels normal to grain dir  [mm]
    e1 = tSetup[specimen]['e1']               # End distance from dowel in grain dir         [mm]
    e2 = tSetup[specimen]['e2']               # Side distance from dowel normal to grain dir [mm]
    n_row    = tSetup[specimen]['n_row']      # Number of rows of dowels    []
    n_col    = tSetup[specimen]['n_col']      # Number of colons of dowels []

    t1 = 2.0                                  # Thickness of ring 1(inner ring) around the dowels [mm]
    t2 = 3.0                                  # Thickness of ring 2(outer ring) around the dowels [mm]

    timberGeom = (b, h, l, d, a1, a2, e1, e2, t1, t2, n_row, n_col, opening_angle)

    # Timber material model
    E11  = 10000                # E-modulus for timber       [N/mm^2]
    E22  = 800                  # E-modulus for timber       [N/mm^2]
    E33  = 400                  # E-modulus for timber       [N/mm^2]
    nu23 = 0.6                  # poisson's ratio
    nu13 = 0.6                  # poisson's ratio
    nu12 = 0.5                  # poisson's ratio
    G23  = 30                   # Shear modulus              [N/mm^2]
    G13  = 600                  # Shear modulus              [N/mm^2]
    G12  = 600                  # Shear modulus              [N/mm^2]

    timberMat = (E11, E22, E33, nu23, nu13, nu12, G23, G13, G12)

    if specimen == 'S1' or specimen == 'S2':
        E_list = [787, 383, 391.3]  # Fitted E-modulus for ring 1 [N/mm^2]
        E_r1 = E_list[n_row-1]
    elif specimen == 'S3':
        E_r1 = 262 - 69*n_row       # Fitted E-modulus for ring 1 [N/mm^2]
    else:
        E_r1 = 500              # E-modulus for ring 1       [N/mm^2]
    nu_r1    = 0.3              # nu for ring 1              [-]
    adj_E_r2 = 1.0              # Adjustment factor for stiffness
    adj_G_r2 = 1.0              # Adjustment factor for shear modulus

    ringMat = (E_r1,nu_r1,adj_E_r2,adj_G_r2)
    # Create a dictionary that stores the plate-setup for S1,S2,S3, Dorn and manual setup
    plSetup = {
        'S1' : {
            'tPl'    : 10.0  ,                  # Thickness of steel plates                     [mm]
            'tL'     : 240.0 ,                  # Length of steel plates in timber              [mm]
            'plLOut' : 50.0  ,                  # Length of steel plates out of timber          [mm]
            'plW'    : 150.0 ,                  # Width of steel plates                         [mm]
            'a1Pl'   : 60.0  ,                  # Distance between dowels in load direction     [mm]
            'a2Pl'   : 55.0  ,                  # Distance between dowels normal load direction [mm]
            'e1Pl'   : 20.0  ,                  # Distance between dowels and plate loaded end  [mm]
            'e2Pl'   : 20.0  ,                  # Distance between dowels and plate side edge   [mm]
            'plGap'  : 1.5   ,                  # Gap between plate and timber                  [mm]
            'plCut'  : (2.5)*a1+e1,             # Height of the cut out in the outer timberpart [mm]
        },
        'S2' : {
            'tPl'    : 10.0  ,                  # Thickness of steel plates                     [mm]
            'tL'     : 240.0 ,                  # Length of steel plates in timber              [mm]
            'plLOut' : 75.0  ,                  # Length of steel plates out of timber          [mm]
            'plW'    : 150.0 ,                  # Width of steel plates                         [mm]
            'a1Pl'   : 60.0  ,                  # Distance between dowels in load direction     [mm]
            'a2Pl'   : 55.0  ,                  # Distance between dowels normal load direction [mm]
            'e1Pl'   : 20.0  ,                  # Distance between dowels and plate loaded end  [mm]
            'e2Pl'   : 20.0  ,                  # Distance between dowels and plate side edge   [mm]
            'plGap'  : 1.5   ,                  # Gap between plate and timber                  [mm]
            'plCut'  : (2.5)*a1+e1,             # Height of the cut out in the outer timberpart [mm]
        },
        'S3' : {
            'tPl'    : 10.0  ,                  # Thickness of steel plates                     [mm]
            'tL'     : 225.0 ,                  # Length of steel plates in timber              [mm]
            'plLOut' : 40.0  ,                  # Length of steel plates out of timber          [mm]
            'plW'    : 160.0 ,                  # Width of steel plates                         [mm]
            'a1Pl'   : 55.0  ,                  # Distance between dowels in load direction     [mm]
            'a2Pl'   : 60.0  ,                  # Distance between dowels normal load direction [mm]
            'e1Pl'   : 20.0  ,                  # Distance between dowels and plate loaded end  [mm]
            'e2Pl'   : 20.0  ,                  # Distance between dowels and plate side edge   [mm]
            'plGap'  : 1.5   ,                  # Gap between plate and timber                  [mm]
            'plCut'  : (2.5)*a1+e1,             # Height of the cut out in the outer timberpart [mm]
        },
        'Manual' : {
            'tPl'    : 10.0  ,                  # Thickness of steel plates                     [mm]
            'tL'     : (n_row-1)*a1+e1+20 ,     # Length of steel plates in timber              [mm]
            'plLOut' : 75.0  ,                  # Length of steel plates out of timber          [mm]
            'plW'    : 150   ,                  # Width of steel plates                         [mm]
            'a1Pl'   : 60.0  ,                  # Distance between dowels in load direction     [mm]
            'a2Pl'   : 55.0  ,                  # Distance between dowels normal load direction [mm]
            'e1Pl'   : 20.0  ,                  # Distance between dowels and plate loaded end  [mm]
            'e2Pl'   : 20.0  ,                  # Distance between dowels and plate side edge   [mm]
            'plGap'  : 1.5   ,                  # Gap between plate and timber                  [mm]
```

```
            'plCut'  : (n_row-0.5)*a1+e1+5,       # Height of the cut out in the outer timberpart [mm]
        },
        'Dorn' : {
            'tPl'    : 8.0   ,                    # Thickness of steel plates                     [mm]
            'tL'     : 114 ,                      # Length of steel plates in timber              [mm]
            'plLOut' : 136  ,                     # Length of steel plates out of timber          [mm]
            'plW'    : 72  ,                      # Width of steel plates                         [mm]
            'a1Pl'   : 60.0  ,                    # Distance between dowels in load direction     [mm]
            'a2Pl'   : 55.0  ,                    # Distance between dowels normal load direction [mm]
            'e1Pl'   : 30.0  ,                    # Distance between dowels and plate loaded end  [mm]
            'e2Pl'   : 50.0  ,                    # Distance between dowels and plate side edge   [mm]
            'plGap'  : 1.0   ,                    # Gap between plate and timber                  [mm]
            'plCut'  : (n_row-0.5)*a1+e1+10,      # Height of the cut out in the outer timberpart [mm]
        },
    }

    # Steel plates
    tPl       = plSetup[specimen]['tPl']          # Thickness of steel plates                     [mm]
    tL        = plSetup[specimen]['tL']           # Length of steel plates in timber              [mm]
    plLOut    = plSetup[specimen]['plLOut']       # Length of steel plates out of timber          [mm]
    plW       = plSetup[specimen]['plW']          # Width of steel plates                         [mm]
    a1Pl      = plSetup[specimen]['a1Pl']         # Distance between dowels in load direction     [mm]
    a2Pl      = plSetup[specimen]['a2Pl']         # Distance between dowels normal load direction [mm]
    e1Pl      = plSetup[specimen]['e1Pl']         # Distance between dowels and plate loaded end  [mm]
    e2Pl      = plSetup[specimen]['e2Pl']         # Distance between dowels and plate side edge   [mm]
    plateGap  = plSetup[specimen]['plGap']        # Gap between plate and timber                  [mm]
    plCutOut  = plSetup[specimen]['plCut']        # Height of the cut out in the outer timberpart [mm]
    plateGeom = (tPl,tL,plLOut,plW,a1Pl,a2Pl,e1Pl,e2Pl,plateGap, plCutOut)

    # Mesh controls

    if meshS == 'coarse':
        # coarse mesh
        meshOTP    = 10             # Mesh size in              [mm]
        meshTP     = 5              # Mesh size in              [mm]
        meshDowel = 1.8             # Mesh size in              [mm]
        meshRing1 = 2               # Mesh size in              [mm]
        meshRing2 = 2.2             # Mesh size in              [mm]
        meshStPl   = 4              # Mesh size in              [mm]
    else:
        # fine mesh
        meshOTP    = 10             # Mesh size in              [mm]
        meshTP     = 3              # Mesh size in              [mm]
        meshDowel = 0.6             # Mesh size in              [mm]
        meshRing1 = 0.8             # Mesh size in              [mm]
        meshRing2 = 1               # Mesh size in              [mm]
        meshStPl   = 4              # Mesh size in              [mm]

    meshSize  = (meshOTP, meshTP, meshDowel, meshRing1, meshRing2, meshStPl)

    # Material properties steel
    E_steel    = 210000         # E-modulus for steel         [N/mm^2]
    nu_steel   = 0.3            # Nu for steel                [-]
    fu_steel   = 916            # Ultimate yield-strength     [N/mm^2]
    fu_pl_steel = 470           # Ultimate yield-strength in range (470-630 for small scale tests)    [N/mm^2]

    dowelMat = (E_steel, nu_steel, fu_steel, fu_pl_steel)

    # Loading and measured stiffnesses
    row, col = config[0], config[1]
    if specimen == 'S1':
        F_max  = [9.2, 13.6, 19.6]      # Load in kN, 40% of F_est
        K_m_   = [28.77, 17.29, 17.42] #kN/mm measured by Frette et. al.
        forcey      = F_max[config[0]-1]*(1000/2)      # Load in  [N]
        K_m         = K_m_[config[0]-1]
    elif specimen == 'S2':
        F_max  = [[8.8, 17.6, 26.4],[12.8, 26.8, 40.8],[18.4, 37.2, 56.0]] # Load in kN, 40% of F_est
        K_m_   = [19.04, 14.19, 12.12] #kN/mm measured by Frette et. al. for full rows 1,2,3
        forcey      = F_max[row-1][col-1]*(1000/2)      # Load in  [N]
        K_m         = K_m_[config[0]-1]
    elif specimen == 'S3':
        F_max  = [[7.2, 10.0 , 12.4],[14.0,22.0,30.0],[21.2,26.0,31.2] ]  # Load in kN, 40% of F_est
        K_m_   = [5.27, 7.72 , 6.02 ] #kN/mm measured by Frette et. al. for full rows 1,2,3
        forcey      = F_max[row-1][col-1]*(1000/2)      # Load in  [N] half force due to modelling symmetry
        K_m         = K_m_[config[0]-1]
    else:
        # If you run a manual entered geometry the force specified here will be applied
        n_ef = min(row,row**(0.9)*(a1/(13*d))**(0.5))

        forcey = 13122*n_ef*0.4              # Load in [N] 40% of Fest for S1 timber geometry with 1-10 dowels
        K_m = None

    return timberGeom, timberMat, ringMat, plateGeom, meshSize, dowelMat, forcey, K_m


def connection_model(timberSetup='Manual', configuration=[1,1], opening_angle=180, dowelModelling='2Ring', meshS='coarse' ,analysisType
= 'ForceDisp', run_analysis=False, iteratives=None, Mesh=None):
    """ timberSetup is controlled with 'S1', 'S2' or 'Dorn' otherwise the manually entered geometry is used.
        The configuration is default set to 1 row and 1 column, for 'S1' and 'S2' number of rows can be changed.
```

```python
        The opening angle defines the angle of "contact" between dowel and wood, 180, 90, 45 and 22.5 is allowed.
        Analysis is default set to false to avoid starting analysis by mistake while testing the rest of the code.
    """
    # Reset the model
    Mdb()
    model = mdb.models['Model-1']
    job_name  = timberSetup+'_'+dowelModelling+'_'+str(int(opening_angle))+'_r'+str(configuration[0])+'-c'+str(configuration[1])+'_'+analysisType

    # Input parameters
    timberGeom, timberMat, ringMat, plateGeom, meshSize, dowelMat, forcey, K_m = input_parameters(timberSetup, configuration, opening_angle, meshS)

    if iteratives is not None:
        ringMat = iteratives
    if Mesh is not None:
        meshSize = Mesh
    # Create the geometry for the timber part of the connection and mesh the part
    parts = make_geometry(model,timberGeom ,plateGeom,meshSize,timberSetup, dowelModelling)
    # Define material and assign sections
    make_sections(model, parts, dowelMat, ringMat, timberMat, timberSetup, dowelModelling)
    # Run the assembly function
    aParts = make_assembly(model, parts, timberGeom, plateGeom, timberSetup, dowelModelling)
    # Create boundaries and load
    make_boundaries(model, aParts, forcey, TOL, timberGeom, plateGeom, timberSetup, dowelModelling,analysisType)
    if run_analysis ==True:
        # Run the analysis of the model
        run_model(model, job_name)
        # View the plots in the results
        if iteratives is None and Mesh is None:
            evaluate_results(job_name, configuration, timberSetup)
        # Evaluate the history output request
        if analysisType == 'ForceDisp':
            force = evaluate_historyOutput(job_name, forcey, ringMat, opening_angle, configuration, timberSetup, analysisType, dowelModelling)
        else:
            res_i, modelProperties = evaluate_historyOutput(job_name, forcey, ringMat, opening_angle, configuration, timberSetup, analysisType, dowelModelling)
    else:
        res_i=None
    n_row  = timberGeom[10]
    n_col  = timberGeom[11]
    dowel_config = [n_row*n_col, n_row, n_col]
    if iteratives is None and Mesh is None:
        if run_analysis==True:
            if analysisType == 'ForceDisp':
                return force
            else:
                return res_i,modelProperties
        else:
            return
    elif Mesh is not None:
        return modelProperties
    else:
        return model, res_i, dowel_config, K_m, modelProperties

# define the true objective function for 3rd degree to analyze the data for iterative analysis
def objective3(x, a, b, c, d):
    return a * x + b * x**2 + c*x**3 + d
# define the true objective function for 2nd degree
def objective2(x, a, b, c):
    return a * x + b * x**2 + c
# define the true objective function for 1st degree
def objective1(x, a, b):
    return a * x + b

def angle_study(timberSetup='S1', configuration=[1,1], r1_range=[400,2000,200], a_range=[90.0, 180.0, 90.0]):
    """ Choose what variable range we want to study. In order to see the stiffness implications.
        timberSetup is controlled with 'S1', 'S2', 'Dorn' or manually entered geometry.
        The configuration is default set to 1 row and 1 column, for 'S1' and 'S2' only the number of rows can be changed.

    Args:
        r1_range (list, optional): Start value, end value, step [MPa]. Defaults to [400,1000,200].
        a_range (list, optional): Start value, end value, step [deg]. Defaults to [90.0, 180.0, 90.0].
    """
    job_name = 'varAngle'+str(timberSetup)+'_'+str(configuration[0])+'-'+str(configuration[1])

    # Define variables that will be iterated over
    E_list     = np.arange(r1_range[0],r1_range[1],r1_range[2])
    A_range    = np.arange(a_range[0],a_range[1],a_range[2])

    res_array = np.empty([1, 10])
    row_n     = res_array.shape[0]
    nu_r1     = 0.3
    adj_G_r2  = 1
    adj_E_r2  = 1
    for opening_angle in A_range:
        for Er1 in E_list:
            ringMat = (Er1,nu_r1,adj_E_r2,adj_G_r2)
```

```
            model, res_i, dowel_config, K_m, modelProperties =
connection_model(timberSetup,configuration,opening_angle,'2Ring','coarse','stiffness',True,ringMat)
            res_array = np.insert(res_array,row_n,[res_i], axis = 0)            #
(timberSetup,configuration,opening_angle,'2Ring','coarse','stiffness',True,ringMat)

    res_array = res_array[1:,:]
    print(res_array)
    n_dowels  = dowel_config[0]
    n_row     = dowel_config[1]
    n_col     = dowel_config[2]

    user = getpass.getuser()
    if not os.path.exists('C:\Users\\'+user+'\OneDrive - NTNU\Documents\Results abaqus\\PlotAbaqus'):
            os.makedirs('C:\Users\\'+user+'\OneDrive - NTNU\Documents\Results abaqus\\PlotAbaqus')
    outpath = 'C:\Users\\'+user+'\OneDrive - NTNU\Documents\Results abaqus\\PlotAbaqus\\'
    # Save the results
    np.savetxt(outpath+str(job_name)+'.csv', res_array, delimiter=",")

    #  Plotting the data angle vs connection stiffness
    fig1, axs = plt.subplots(1,1 ,dpi=600)
    outpath1 = outpath+'/'+str(job_name)+'_E1'+'.png'
    ax = axs
    e1Length = len(E_list)
    sInd = 0
    endInd = e1Length
    step_size = e1Length

    for i, value in enumerate(E_list):
        x = res_array[sInd::step_size,7]
        y = res_array[sInd::step_size,4]
        ax.plot(x, y,marker='o', label = 'K for r1 stiffness E=%1.1f'%res_array[sInd,8], zorder=2)
        sInd   += 1

    ax.grid(zorder=1)
    ax.set_xlabel('Contact angle [deg]')
    ax.set_ylabel('Stiffness [kN/mm]')
    ax.legend(loc = 'upper left', fontsize=8)
    fig1.savefig(outpath1)
    plt.close(fig1)

    #  Plotting the data
    fig2, ax = plt.subplots(1,1 ,dpi=600)
    e1Length = len(E_list)
    sInd = 0
    endInd = e1Length
    step_size = e1Length

    for i, value in enumerate(E_list):
        x = res_array[sInd::step_size,4]
        y = res_array[sInd::step_size,7]
        ax.plot(x, y,marker='o', label = 'K for r1 stiffness E=%1.1f'%res_array[sInd,8], zorder=2)
        sInd   += 1

    outpath2 = outpath+'/'+str(job_name)+'_E2'+'.png'
    ax.grid(zorder=1)
    ax.set_ylabel('Contact angle [deg]')
    ax.set_xlabel('Stiffness [kN/mm]')
    ax.legend( fontsize=8) #loc = 'upper left',
    fig2.savefig(outpath2)
    plt.close(fig2)

    #  Plotting the data
    fig3, axs = plt.subplots(1,1 ,dpi=600)
    outpath3 = outpath+'/'+str(job_name)+'_E1_perDowel'+'.png'
    ax = axs
    e1Length = len(E_list)
    sInd = 0
    endInd = e1Length
    step_size = e1Length

    for i, value in enumerate(E_list):
        x = res_array[sInd::step_size,7]
        y = res_array[sInd::step_size,3]
        ax.plot(x, y,marker='o', label = 'K per dowel for r1 stiffness E=%1.1f'%res_array[sInd,8], zorder=2)
        sInd   += 1

    ax.grid(zorder=1)
    ax.set_xlabel('Contact angle [deg]')
    ax.set_ylabel('Stiffness [kN/mm]')
    ax.legend(loc = 'upper left', fontsize=8)
    fig3.savefig(outpath3)
    plt.close(fig3)

    if timberSetup in ('S1','S2'):
        #  Plotting the data per dowel and with a intersection-line
        fig4, ax = plt.subplots(1,1 ,dpi=600)
        fig4.suptitle('Setup with %d rows and %d columns'%(n_row,n_col), fontsize=14)
        outpath4 = outpath+'/'+str(job_name)+'_E1_perDowel'+'.png'

```

```python
        e1Length = len(E_list)
        sInd = 0
        endInd = e1Length
        for i in A_range:
            # Find the correct stiffness data, normalize per dowel and plot
            x = res_array[sInd::step_size,7]
            y = res_array[sInd::step_size,3]
            ax.plot(x, y,marker='o', label = 'R1 K per dowel for E=%1.1f MPa'%res_array[sInd,0], zorder=2)

            # curve fit to the data
            popt, _ = curve_fit(objective3, x, y)
            # popt, _ = curve_fit(objective2, x, y)
            # summarize the parameter values
            a, b, c, d = popt
            # a, b, c = popt
            print('y = %.7f * x + %.7f * x^2 + %.7f*x^3 + %.7f ' % (a, b, c, d))
            # print('y = %.7f * x + %.7f * x^2 + %.7f ' % (a, b, c))
            # define a sequence of inputs between the smallest and largest known inputs
            x_line = np.arange(min(x), max(x), 1)
            g = np.ones(len(x_line))*K_m

            # calculate the output for the range
            y_line = objective3(x_line, a, b, c, d)
            # y_line = objective2(x_line, a, b, c)
            # create a line plot for the mapping function
            ax.plot(x_line, y_line, '--', label = 'Fitted curve E = %1.1f MPa'%res_array[sInd,8] )

            if K_m > min(y_line)  and K_m < max(y_line):

                #Plot the intersection point and print the corresponding E-modulus
                idx = np.argwhere(np.diff(np.sign(g - y_line))).flatten()
                plt.plot(x_line[idx], y_line[idx], 'ro')

                ax.text(1.01*min(x), 0.9*K_m, ('Angle = %.2f deg'%x_line[idx]) , fontsize=10)
                sInd   += 1
            else:
                raise ValueError('Measured stiffness is not within the interval')

        ax.plot(x_line, g, '-', label = 'Measured stiffness')
        ax.grid(zorder=1)
        ax.set_xlabel('Angle [deg]')
        ax.set_ylabel('Stiffness [kN/mm]')
        ax.legend( fontsize=6)
        fig4.savefig(outpath4)
        plt.close(fig4)
    return

def variable_study(timberSetup='Manual', configuration=[1,1], opening_angle=180, r1_range=[400,1000,200], r2_range=[1, 1.5, 0.5]):
    """ Choose what variable range we want to study. In order to see the stiffness implications.
        timberSetup is controlled with 'S1', 'S2' or 'Dorn' otherwise the manually entered geometry is used.
        The configuration is default set to 1 row and 1 column, for 'S1' and 'S2' only the number of rows can be changed.

    Args:
        r1_range (list, optional): Start value, end value, step [MPa]. Defaults to [400,1000,200].
        r2_range (list, optional): Start value, end value, step [multiplier]. Defaults to [1, 1.5, 0.5].
    """
    job_name = 'iterationE'+str(timberSetup)+'_'+str(configuration[0])+'-'+str(configuration[1])
    # Define variables that will be iterated over
    E_list    = np.arange(r1_range[0],r1_range[1],r1_range[2])
    E_r2list  = np.arange(r2_range[0],r2_range[1],r2_range[2])

    res_array = np.empty([1, 10])
    row_n     = res_array.shape[0]
    nu_r1     = 0.3
    adj_G_r2  = 1
    for adj_E_r2 in E_r2list:
        for Er1 in E_list:
            ringMat = (Er1,nu_r1,adj_E_r2,adj_G_r2)
            model, res_i, dowel_config, K_m, modelProperties =
connection_model(timberSetup,configuration,opening_angle,'2Ring','coarse','stiffness',True,ringMat)
            res_array = np.insert(res_array,row_n,[res_i], axis = 0)

    res_array = res_array[1:,:]
    print(res_array)
    n_dowels  = dowel_config[0]
    n_row     = dowel_config[1]
    n_col     = dowel_config[2]


    user = getpass.getuser()
    if not os.path.exists('C:\Users\\'+user+'\OneDrive - NTNU\Documents\Results abaqus\\PlotAbaqus'):
            os.makedirs('C:\Users\\'+user+'\OneDrive - NTNU\Documents\Results abaqus\\PlotAbaqus')
    pathTosave = 'C:\Users\\'+user+'\OneDrive - NTNU\Documents\Results abaqus\\PlotAbaqus\\'

    # Save the results
    np.savetxt(pathTosave+str(job_name)+'.csv', res_array, delimiter=",")

    #  Plotting the data
    fig1, axs = plt.subplots(1,1 ,dpi=600)
```

```python
    outpath1  = pathTosave+str(job_name)+'_E1'+'.png'
    ax = axs
    e1Length = len(E_list)
    sInd = 0
    endInd = e1Length
    for i, value in enumerate(E_r2list):
        x = res_array[sInd:endInd,8]
        y = res_array[sInd:endInd,4]
        ax.plot(x, y,marker='o', label = 'R1 K for E_r2=%1.1f*E_wood'%res_array[sInd,1], zorder=2)
        sInd   += e1Length
        endInd += e1Length
    ax.grid(zorder=1)
    ax.set_xlabel('E-modulus [MPa]')
    ax.set_ylabel('Stiffness [kN/mm]')
    ax.legend(loc = 'upper left', fontsize=8)
    fig1.savefig(outpath1)
    plt.close(fig1)

    #  Plotting the data
    fig2, ax = plt.subplots(1,1 ,dpi=600)
    e2Length = len(E_r2list)
    sInd = 0
    step_size = e1Length

    for i, value in enumerate(E_list):
        x = res_array[sInd::step_size,9]*100
        y = res_array[sInd::step_size,4]
        ax.plot(x, y,marker='o', label = 'R2 K for E_r1=%1.1f MPa'%res_array[sInd,0], zorder=2)
        sInd   += 1

    outpath2 = pathTosave+str(job_name)+'_E2'+'.png'
    ax.grid(zorder=1)
    ax.set_xlabel('E-modulus of wood in %')
    ax.set_ylabel('Stiffness [kN/mm]')
    ax.legend( fontsize=8) #loc = 'upper left',
    fig2.savefig(outpath2)
    plt.close(fig2)

    #  Plotting the data
    fig3, axs = plt.subplots(1,1 ,dpi=600)
    outpath3 = pathTosave+str(job_name)+'_E1_perDowel'+'.png'
    ax = axs
    e1Length = len(E_list)
    sInd = 0
    endInd = e1Length
    for i, value in enumerate(E_r2list):
        x = res_array[sInd:endInd,8]
        y = res_array[sInd:endInd,3]
        ax.plot(x, y,marker='o', label = 'R1 K per dowel for E_r2=%1.1f*E_wood'%res_array[sInd,1], zorder=2)
        sInd   += e1Length
        endInd += e1Length
    ax.grid(zorder=1)
    ax.set_xlabel('E-modulus [MPa]')
    ax.set_ylabel('Stiffness [kN/mm]')
    ax.legend(loc = 'upper left', fontsize=8)
    fig3.savefig(outpath3)
    plt.close(fig3)


    return

def mesh_testing():

    user = getpass.getuser()
    if not os.path.exists('C:\Users\\'+user+'\OneDrive - NTNU\Documents\Results abaqus\\Text'):
            os.makedirs('C:\Users\\'+user+'\OneDrive - NTNU\Documents\Results abaqus\\Text')
    pathTosave = 'C:\Users\\'+user+'\OneDrive - NTNU\Documents\Results abaqus\\Text\\'

    meshSizes = [np.arange(2,11,1),np.arange(2,11,1),np.arange(0.6,2.7,0.3),np.arange(0.6,2.7,0.3),np.arange(0.6,2.7,0.3),np.arange(2,11,1)]
    meshOTP    = 10              # Mesh size in               [mm]
    meshTP     = 5               # Mesh size in               [mm]
    meshDowel  = 1.8             # Mesh size in               [mm]
    meshRing1  = 2               # Mesh size in               [mm]
    meshRing2  = 2.2             # Mesh size in               [mm]
    meshStPl   = 4               # Mesh size in               [mm]
    meshSize   = [ meshOTP, meshTP, meshDowel, meshRing1, meshRing2, meshStPl]
    res_array = np.empty([1,4])
    for i in range(len(meshSize)):
        res_array = np.empty([1,4])
        for mesh in meshSizes[i]:
            meshSize[i] = mesh
            start=time.time()
            mProp = connection_model('S1',[1,1],180,'2Ring','coarse','stiffness',True,None,meshSize)
            tot = time.time()-start
            mProp[3] = tot
            res_array = np.insert(res_array,0,[mProp], axis = 0)
        np.savetxt(pathTosave+str(i+1)+'_mesh_test.csv', res_array, delimiter=",")
        print(res_array)
```

```
    return

def main(analysis_type='N'):
    """ Normal analysis is the default, for a study of the angle choose 'A' and for a study of E-modulus in ring1 and ring2 choose 'E'.
        Some parameters can be set directly as input in the three functions here, other must be set in the input function.

    Args:
        analysis_type (str, optional): 'N'- Normal, 'A' - Angle, 'E' - E-modulus, 'M' - mesh testing. Defaults to 'N'.
    """

    if analysis_type == 'N':    # If only one analysis is to be carried out run the connection model
        connection_model('S1', [3,1],45.0,'2Ring','coarse','stiffness')
    elif analysis_type == 'A':  # If we want to study the contact angle effect
        angle_study('S3',[3,3],[55.0,155.0,100],[22.5,200,22.5])
    elif analysis_type == 'E':  # If we want to study variables run the iteratively formula
        variable_study('S2',[3,1],45,[100,1000,100],[0.5,2.0,0.5])#[200,1100,100],[0.5,2.0,0.5])
    elif analysis_type == 'M':  # If we want to study the mesh size effect on stiffness
        mesh_testing()

    return

##########################################################################################
##                         Run the file to create the model                             ##
##########################################################################################
start_time = time.time()
main('N')
print("--- %s seconds ---" % (time.time() - start_time))

# Adress to run code in Abaqus if the code is in the temp folder:
# Run this: execfile(r'mainscript.py')
# If it is in a different folder, add the path in front of the filename like this:
# execfile(r'C:\Users\GOT VISION\Documents\GitHub\Master-project-Abaqus\mainscript.py')
```

Hattestad and Vangsnes

An Experimental and Numerical Investigation of Stiffness in Dowel-type Connections

NTNU
Norwegian University of
Science and Technology