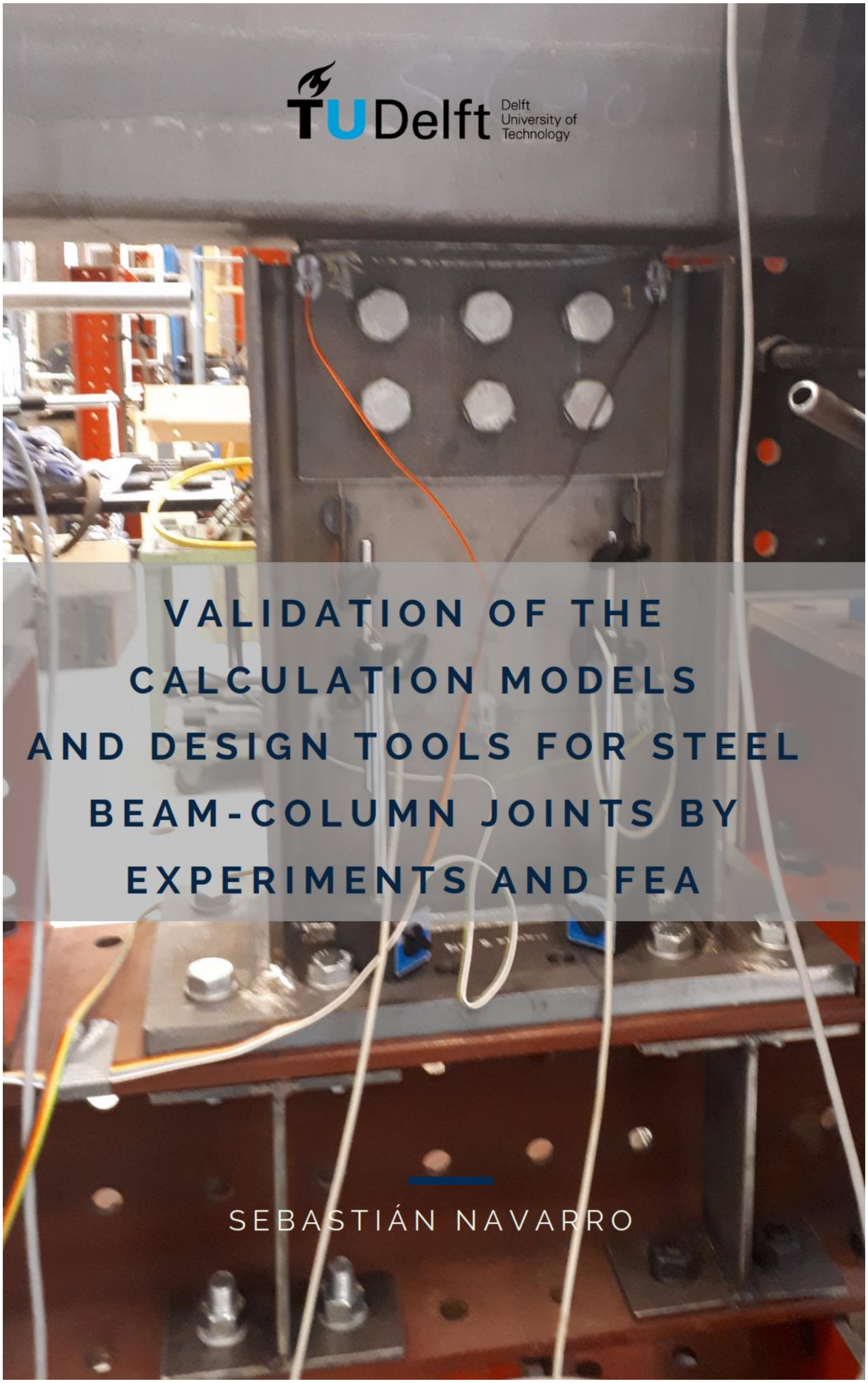


VALIDATION OF THE CALCULATION MODELS AND DESIGN TOOLS FOR STEEL BEAM-COLUMN JOINTS BY EXPERIMENTS AND FEA

SEBASTIÁN NAVARRO

Validation of the Calculation Models and Design Tools for Steel Beam-Column Joints by Experiments and FEA

By

Sebastián Navarro Villagómez

in partial fulfilment of the requirements for the degree of

Master of Science

in Civil Engineering

Track

Structural Engineering

Specialization

Steel and Timber Construction

at the Delft University of Technology,
to be defended publicly on Friday 22, 2017 at 15:00.

Supervisor:	Prof. dr. Milan V. Veljkovic	
Thesis committee:	Dr. Marko Pavlović	TU Delft
	Ir. Antonello Lípori	ASK Romein
	Dr. ir. Max A. N. Hendricks	TU Delft
	Ir. Goran Pavlović	ASK Romein

Preface

The following document is the report of my thesis "Validation of the Calculation Models and Design Tools for Steel Beam-Column Joints by Experiments and FEA". This paper is required as part of my curriculum for my master's degree in Science in Civil Engineering. Its purpose is to contribute and to deepen in the analysis of steel joints. Topics such as hybrid welds, simple joints to flexible elements and especially the use of finite elements in the daily practice of a structural engineer, are approached.

The thesis was a collective project between the TU Delft university and the company ASK Romein, who together wanted to extend the analysis in pioneer details such as high strength welding. In addition, in the last years, steel design software has appeared in the market in order to improve productivity and facilitate the work of structural engineers. These new powerful tools that come from the hands of the world's technological advances must be taken with care and their results must be validated to be able to realize efficient, but above all, safe designs.

I would like to thank TU Delft, my university, where I have done my specialization and where I have achieved my objective of learning and deepening the subject that I am passionate about: the design of steel structures. Within this context, I would like to thank my daily supervisor Marko Pavlović, who not only provided me with his guide and with whom I discussed enough time to make this project a reality, but also to encourage in me the passion for structural design, especially the design in steel, through his lectures and his desire to always carry out challenging, practical and interesting projects. To Slobodanka Jovašević also big thanks, for sharing with me her knowledge and time.

A special thanks to the company ASK Romein, specially to my boss Antonello Lípori, for giving me the opportunity to make a project that includes all the phases of the steel design, from its conception in design to its construction (experiment) going through all areas of drawing, modeling, management and production. Being able to put theory to pass through experiments, to contrast conceptual design with practical reality, is a unique opportunity for which I will always be grateful. A big thank to my team Goran, Ian, Alexander and Andy for their advices, and for making the office a pleasant, fun and positive place.

I would like to thank TU Delft's steel laboratory. Especially to Kevin Mouthaan who gave himself completely to the experiment, making my project his own and with whom I learned a lot from the practice of laboratory work and steel structures. The freedom that the university laboratory gives us to put in practice our ideas and develop our research is the reason why TU Delft is a knowledge incubator and world leader in higher education.

A special thanks to Ecuador and his national government that through its scholarship programs encourages the specialization of professionals. This made possible my studies in one of the best universities in the world.

A big hug and thanks to my family in Ecuador and The Netherlands who always give me their support, trust and love. To my parents for their support and for taking care of the most important thing I have, my daughter. To my friends for making me feel that their affection awaits me intact in Ecuador and my "international" friends in Holland with whom I have shared these two years full of learning, joys and good times.

Finally, I want to dedicate this thesis to my daughter Rafaela. Let this project be an example for her that shows that limits are created by ourselves, that she can reach whatever she wants to and that her goals can be as great as her dreams. Thank you for waiting for me these two years and for giving me your love in spite of any distance. As you said, "everything will be fine when we meet again."

Sebastian Navarro
Delft, August 2017

Abstract

New technologies (like 3D printing) and the increasing use of no conventional materials (like Fiber Reinforcement Polymer - FRP) are changing the construction industry. More effective and efficient structures are needed in order to achieve sustainable projects. Steel construction industry is not outside this modifications and advances. To keep steel construction competitive in the market, the industry needs to adapt to these changes by optimizing its manufacturing and erection processes. In a steel structure, joints determine almost 50% of the total costs (Bijlaard, 2006). This percentage can drastically decrease if some details are avoided, like stiffeners. Nevertheless, the only way to eliminate these components is through a deepest analysis of the joint. This kind of analysis can be slow and expensive hindering the design process. In this context new numerical tools are appearing in the market in order to improve productivity and facilitate the work of structural engineers. These new powerful tools that come from the hands of the world technological advances, must be taken with care and their results must be validated to be able to realize their efficient, but above all, their safety.

In this project, the improvement of the design of steel joints is addressed. This goal was achieved through two major tasks. First, an experiment was performed in order to deepen in the knowledge of simple joints when they are used between "I" shape beams and a flexible element like a hollow section face. The use of high strength welds and its implications were tested in the experiments and at the same time the real behavior of the joint was addressed. The new Eurocode 1993 1-8 (2020) contemplates the use of high strength steel and welds in steel practice. Hybrid welds can lead to a more efficient production process. Secondly, numerical analysis, hand calculations following the Eurocode Rules and previous documented experiments (SERICM II) were compared to validate the use of a finite element tool for daily practice structural engineering. The comparison was made in terms of strength and resistance of the joints and it proved that specialized finite element tools can be used in a safe and efficient way. These tools are able to predict in an accurate manner the joint's resistance according to the Eurocode Rules.

.

Content Table

Table of Figures

Chapter 1 – Introduction

New technologies (like 3D printing) and the increasing use of no conventional materials (like Fiber Reinforcement Polymer - FRP) are changing the construction industry. Fast and efficient (in energy consumption and quantity of raw material terms) solutions are needed in the industry in order to keep track of sustainability and the circular economy approach applied to the building industry. Steel construction industry is not outside this modifications and advances. To keep steel construction competitive in the market, the industry needs to adapt to these changes. In a steel structure, joints determine almost 50% of the total costs (Bijlaard, 2006). This percentage can drastically decrease if some details are avoided, like stiffeners. Nevertheless, the way to eliminate these components is through a deepest analysis and understanding of the joint. In these terms, the current code opens the possibility of designing more efficient joints through new classification categories like partial strength and semi rigid joints. To allow structural engineers to design partial strength or semi-rigid joints, Eurocode EN1993 1-8 (2005) introduces a design process of joints called “Component Method”. Here the joint is divided in its principal components that resist the transfer of internal forces due to external loading. An idealization of the components into (rotational) non-linear springs is made. After defining the strength of each component, the springs are assembled in series and parallel to reproduce the behavior of the joint. The local focus of the method in the components of the joint leads to a better understanding of the joint behavior and at the same time allows a more efficient design. Nevertheless, the method can introduce an important amount of work (increase the design time) and some complexity to the design process. Nonetheless, the method also allows the designer to explore and use different shapes and geometries, as long as the designer is capable of relate the components of his joint with the components given in EN1993 1-8.

The increase in the difficulty and time spent in the steel joint design due to the use of the component method and also due to the limitations of using joints with geometries, beyond the use of the components present in the code (non-standard joints), has created the need to improve the design process of a steel joint. For this reason, the importance of using a numerical method (accompanied by the development of friendly user tools) that presents the possibility of optimizing the design of steel joints and that takes full advantage of the potential of the Component Method and the new classification of joints presented in the code, grows every day.

In this thesis, the improvement of the design of steel joints is addressed. This goal was achieved through two major tasks. First, an experiment was performed in order to deepen in the knowledge of simple joints when they are use between “I” shape beams and a flexible element like a hollow section face. The use of high strength welds and its implications were tested in the experiments and at the same time the real behavior of the joint was addressed. The new Eurocode En1993 1-8 (2020) contemplates the use of high strength steel and welds in steel practice. Hybrid welds can lead to a more efficient production process. Second, numerical analysis, hand calculations following the Eurocode Rules and previous documented experiments (SERICM II) were compared to validate the use of a finite element tool for daily practice structural engineering. The comparison was made in terms of strength and resistance of the joints and proved that specialized finite element tools can be used in a safe and efficient way. These tools are able to predict in an accurate manner the joint resistance according to the Eurocode Rules.

1.1 Document Structure

This master thesis is divided in four main chapters and one main annex. Chapter 2 presents the state of art in steel joints analysis under the concept of finite element modeling. In addition, basic principles are tackle in order to give a background about finite elements analysis, the component method and hybrid welds.

Chapter 3 will present in an extensive manner the preparation, analysis and results of the experiment of a real size steel non-standard steel joint. Chapter 4 studies the impact of the new method of fillet welds design proposed in the new version of the EN 1993 1-8 (2020). A numerical analysis for the simplified proposed model is also performed in this section. Chapter 5 presents the comparison of two numerical tools that uses Finite Element Analysis. The comparison will be in terms of its accuracy and safety prediction of the characteristic and real resistances of three different types of joints. These joints were obtained from SERICOM database.

Annex A gives an analysis of the new software IDEA StatiCa, its theoretical background, the potential benefits of its use and a setup proposal of the software configuration, in order to give to the user an idea of the correct use of the software. Annex B, C and D presents the hand calculations, the tensile coupon tests results and other material properties.

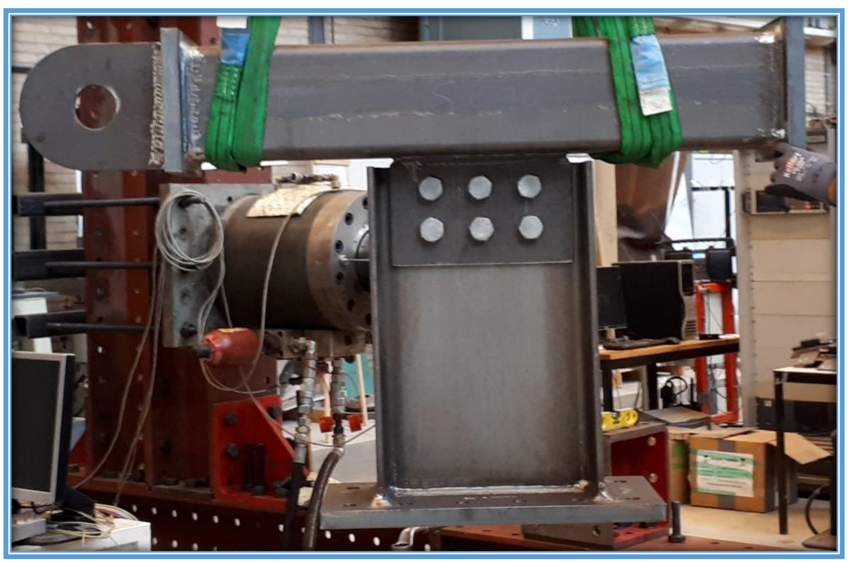


Figure 1- 1: Experiment Specimen (Steel Joint)

Chapter 2 – Literature Review

2.1 Introduction

Joints have a major impact on the global performance of a steel structure. Its idealization for the analysis leads to different approaches when designing structural elements (ex. simple supported beams). However, in order to attain an idealized behavior, important amounts of resources need to be invested, especially to achieve in practice the idealized behavior of a rigid joint. In order to reduce costs in a steel structure, joints need a deeper analysis to understand their behavior and to be able to optimize the details. Moreover, new challenges in this industry are related to more complex shapes and geometries of the structures. The huge variety of shapes and types of joints has created the need to use modern tools of analysis in order to accomplish the expected optimization of the structural detail.

The aim of this Literature Review is to give an insight of three fundamental concepts presented during the progress of the project. These concepts will be used as background in the coming analyses performed in the master thesis. In addition, the state of art of them will be addressed. These three concepts are:

- *Component Method:* is the widely accepted method for steel joints analysis and it is regulated in the current code (EN1993 1-8 (2005). This method opens the possibility for structural engineers to analyze any type of joint as long as they are capable to relate the components in their joints to the basic components presented in the code.

- *Finite Element Analysis:* this numerical tool for the solution of most engineering problems has been used for many years to validate and expand experimental works in the research field for most of the engineer branches. The possibility of using this tool in a friendly interface environment, which is now possible thanks to the introduction of new software in the market, as a daily practice tool will be studied in this master project.

- *Hybrid Welds:* the new version of the code (EN1993 1-8 (2020)) allows the possibility of using High Strength Steel and HSS welds. This opens up the possibility of optimizing welding connections in order to reduce welding costs which are one of the biggest contributors to the overall cost of steel structures.

2.2 Introduction to the Component Method

The detailing of steel joints is one of the most important points in the design of steel structures. Its influence on the structural behavior and the total cost of the structure is of great impact. The analysis of joints goes from global to specific. In terms of global analysis, the idealization of the joints is fundamental for the distribution and transfer of the acting forces from element to element through its connection. This transfer can be continuous, semi-continuous or simple, depending on the ability of the connection to ensure the rotational continuity between the elements. After the analysis, the internal solicitations that the joints are going to be subjected are obtained. It is important to mention that the stiffness used to idealize the joint directly influences the type and magnitude of the load transfer.

When the analysis goes from global to local there are several challenges that must be considered. *“The analysis of the behavior of steel joints is very complex and requires the proper consideration of a multitude of phenomena, ranging from material non-linearity (plasticity, strain-hardening), non-linear contact and slip, geometrical non-linearity (local instability) to residual stress conditions, and complicated geometrical configurations*”(Simões da Silva, 2008). For this reason, the so-called "Component Method" is today the accepted method for the analysis of steel joints. This method consists on identifying and separating the different components that conform the joint to be able to analyze them in an isolated way. These components must be related to the "basic components" that appear in the code EN1993 1-8. In the regulation, the basic components are identified and the formulation to obtain their strength and stiffness is already established. In this way, the component method can be used for any geometric configuration of a joint (standard and non-standard joints) as long as the designer is able to relate them to the basic components of the code (Bijlaard, 2006). The structural behavior of the joint must be calculated by assembling the components, which are idealized as springs, which are placed in series or parallel to describe the overall behavior of the joint.

Jaspar in his guide for the design of steel joints describes three steps for the use of the component method (Jaspart and Weynand, 2016):

1) *Identification of the active components. T*he first step is to separate the joint in components and, from there, define which are the active ones in the specific analyzed situation. The code and its user guides suggest that this selection should be made on the basis of a logical assumption of the joint's behavior. It is necessary to take into account a real distribution of the internal forces and the compatibility of the deformations based on the stiffness of the components. An example of this is one type of hybrid steel joints where the mix of bolts and welds are present. Here it should be established how real is the participation of the bolts when the welding is present due to the higher stiffness of the weld compared to the bolts.

2) *Individual Assessment of the Stiffness and Characteristic Strength of the Components.* Once the active components have been identified and related to the basic components present in the code, their characteristic strength and stiffness should be evaluated using the formulas present in the code. In case of identifying a component that is not present in the code, the design per experiment, stipulated in EN1990 can be applied.

3) *Joint Assembly.* Once the individual resistances and stiffness values of each component are calculated, the components are idealized in springs that when located in parallel or in series determine the behavior of the joint. This is how the strength and stiffness of the joint are calculated as a whole.

In Annex E, table 6.1 of the current code (EN1993 1-8 (2005)) is presented. This table contains the basic components that are in the code. The new version of the code (EN1993 1-8 (2020)) provides this information in its "Annex A - Description and Formulation of the Basic Components".

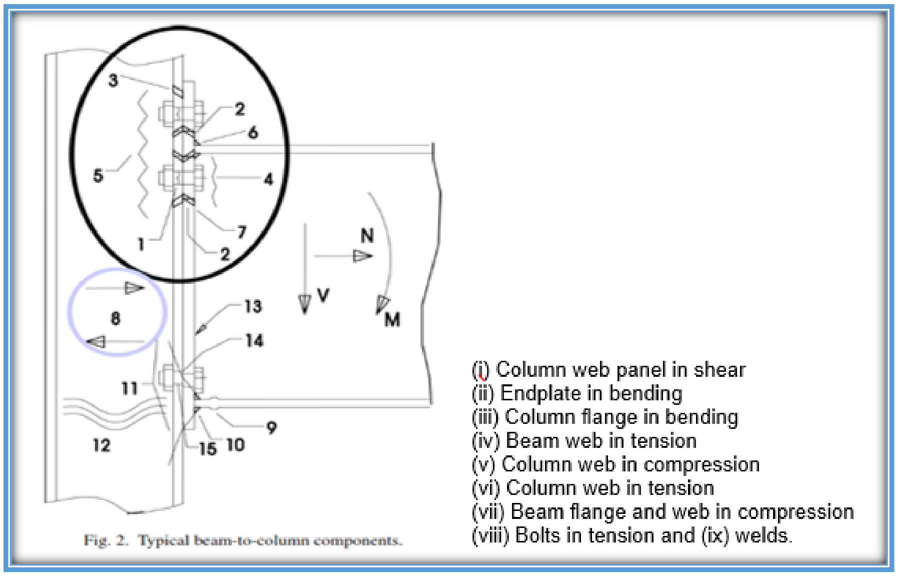


Figure 2 - 1: Typical Components from a Column-Beam Joint (Simões da Silva, 2008)

2.3 Joint Resistance Calculation

In the calculation of steel joints strength, it can be identified two groups that have their own design method. In one hand, the Rigid and Semi-Rigid joints that transfer considerable bending moments use the component method. On the other hand, simple joints use a different approach. For this group the failure modes analysis is used. The calculation of the resistance for these two groups is explained in more detailed below.

2.3.1) Simple Joints

For simple joints it is assumed that there is no bending moment transfer. For this reason, its design and detailing should be defined in such a way that the joint avoids such transfer. The code assumes that these types of joints are designed for shear or axial loads only. However, in *“Eurocode 3 Part 1-8, no specific design rules are provided for simple joints"* (Jaspart and Weynand, 2016). As consequence, authors such as (Jaspart et al, 2009), (Weynand et al, 2013) and (Moreno et al, 2011) have developed and combined different regulations and recommendations for the proper analysis of simple joints. For future versions of the code, it is intended the implementation of a general guide like is already present in the "European Design Recommendations for the Design of Simple Joints in Steel Structures" (Jaspart et al, 2009).

The calculation method for simple joints is through an analysis of all possible failures modes happening in the joint. In this way, each mode is evaluated and its pre-failure resistance is determined. The lower resistance between all the modes determines the final strength of the joint. This method can be compared to a string. In principle, the string will be broken in the weakest link. By determining the resistance of each link and identifying the weakest, the resistance of the whole string is obtained.

There are three types of simple joints: Header Plate, Fin Plate and Web Cleat (see Figure 2-2). (Jaspart and Weynand, 2016) presents a design guide for each of them from where it is possible to emphasize two fundamental characteristics to be able to affirm that a joint can produce a plastic hinge, and those are:

a) Joint possess a sufficient rotation capacity
b) Joint possess a sufficient ductility

Fulfilling these two principles ensure and certify the assumption of perfect hinge for simple joints idealization. The rotational capacity is linked to the ability of the joint to deform without generating too much resistance to internal bending moments. The second principle is more related to the type of failure mode that is expected in the analysis. On the one hand, if failure on the end plate is present, it can be say that the joint have a ductile behavior. However, if the failure in weld or bolt is predicted, a brittle performance will occur in the joint.

An example of the design of such joints can be seen in Annex B, where the design of the experiment joint (simple) is shown.

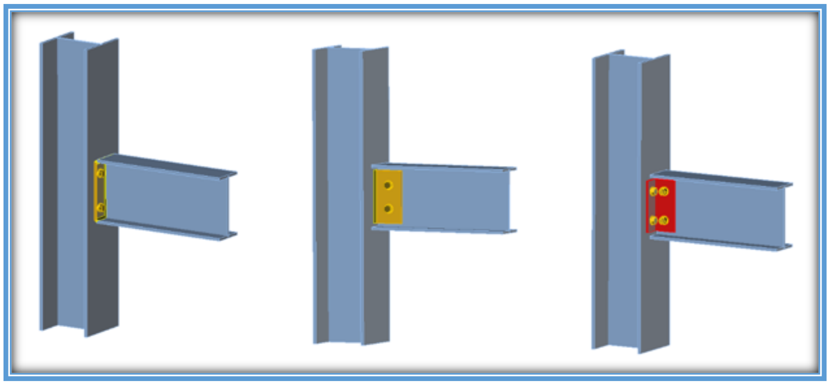


Figure 2 - 2: Types of Simple Joints

2.3.2) Component Method – "T-Sub"

As explained in 2.2), the calculation of the strength of a moment-resistant joint should be performed under the component method. The assembly of components is, perhaps, where the designer should pay most of its attention since the assembly of the springs of each component must follow the physical principles of transfer of forces and at the same time the compatibility of deformations must be ensured. The flow of the internal forces can follow an Elastic or Plastic distribution according to the capability of each element to produce ductile behavior.

The following image (figure 2-3) shows the flow-chart for the assembly of the connections presented by (Jaspart and Weynand, 2016). An example of the use of the component method can be seen in Annex D, which presents the calculation of rigid and semi-rigid joints used in Chapter 5.

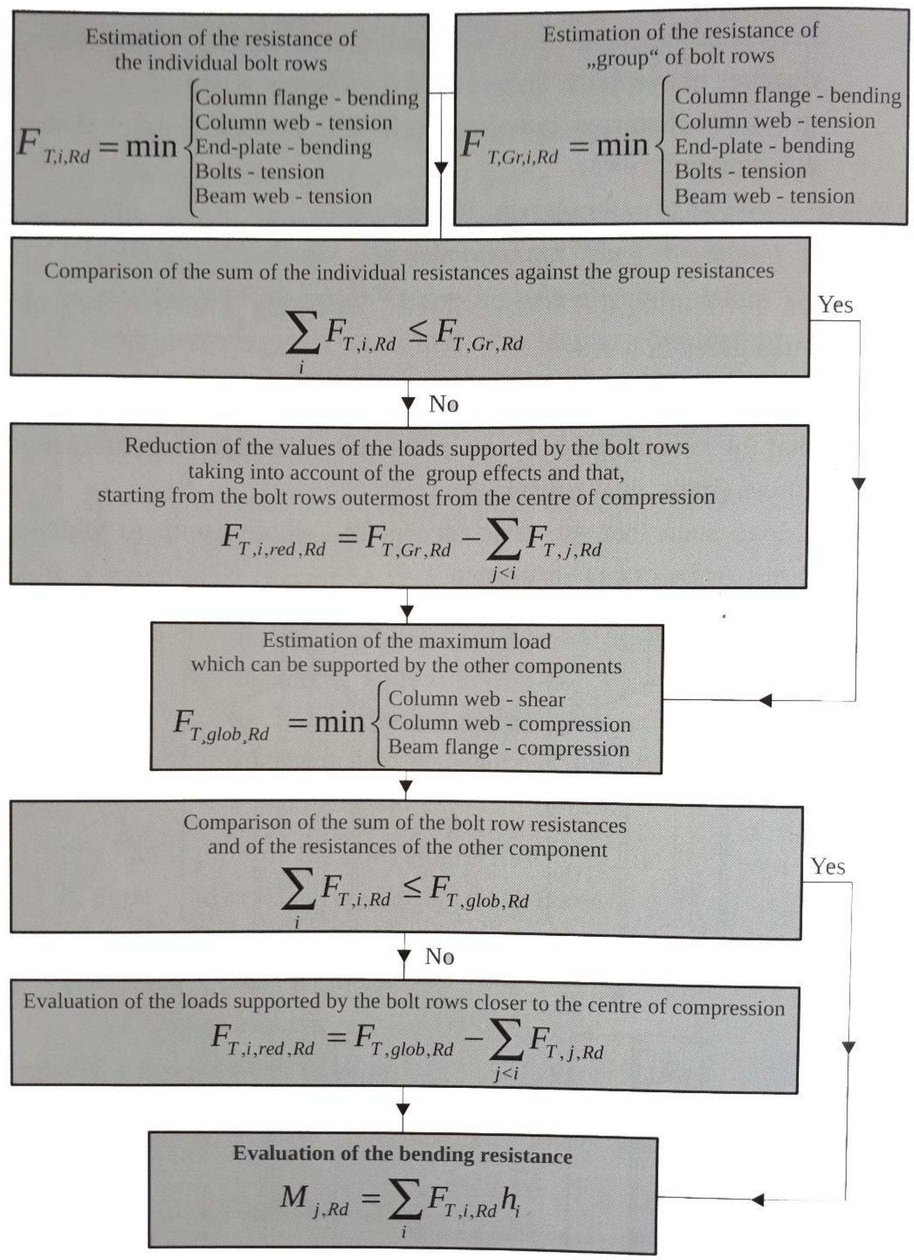


Figure 6.19 – Flow-chart for the assembly procedure

Figure 2 - 3: Assembly Flow-Chart (Jaspart and Weynand, 2016)

Among all the basic components found in the code, those using the "T-sub" idealization are probably the most influential and complex (accompanied by the shear panel of the column). The T-Sub idealization is the method of analyzing components that transfer forces through bending. When two elements are connected using bolts in their flanges, the behavior of the connection can be assimilated to the T-Sub component (see figure 2-4). T-Sub is subjected to tension load and its performance is closely linked to the geometry of the joint and the properties of the materials in them. When these types of components fail, three types of failure modes can be observed.

1. *Mode 1:* When the plate fails by bending. In this mode the deformation and resistance of the plate, which can be idealized as a beam with four plastic hinges, is the one that determines the failure. This mode should be preferred by the designer since it is highly ductile. In places where seismic loads exist, this mode of failure is mandatory.

2. *Mode 2:* In this mode the bending failure of the plate is combined with tension failure of the bolts. Although it may show some ductility, the breakage of the bolts is abrupt

3. *Mode 3:* It is the failure of the joint due to tension in the bolts. This is a brittle failure.

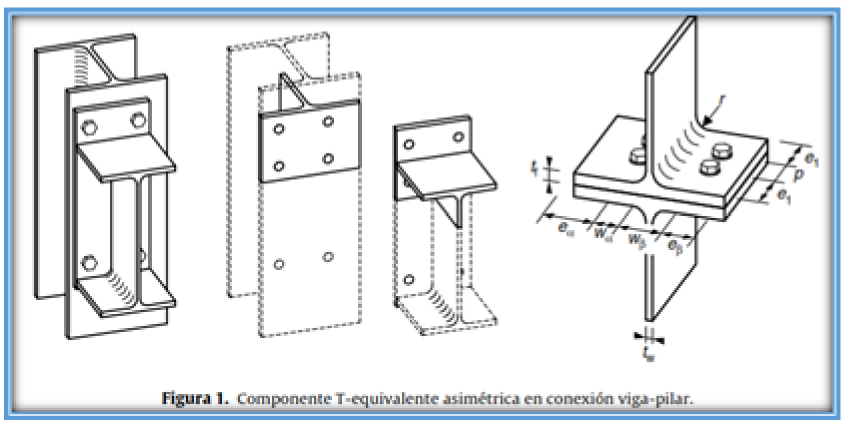


Figure 2 - 4: T-Sub Idealization (Jiménez de Cisneros et al., 2017)

2.4 Joint Classification (EN1993 1-8 (2005))

In this section the two types of classification for steel joints stipulated in Eurocode 3 is presented. An accurate classification is important at the time of structural analysis because the assumptions made in the global analysis must be reflected in the local design of each joint. For example, if a simply supported beam is designed and, after the classification of the joint, is determined to be a semi-rigid joint, this will have consequences on the actual distribution of internal forces and deformation in the beam. On one hand the semi-rigid joint will give less deformation than the assumed as pinned joint, but the presence of negative moments near the ends of the beam can lead to the failure of it, or at least cracks in the floor system.

The code classifies steel joints in two ways: due to their stiffness and its resistance.

2.4.1) Classification by Stiffness

When a joint is classified by its stiffness it can be placed in three categories: rigid, semi-rigid and nominally pinned. The introduction of the semi-rigid category by the code allows a more efficient design not only of the joint but also of the structure itself. The main parameters of the three categories are detailed below:

a. *Rigid Joints:* In order for a joint to fall into this category, it must ensure fully rotational stiffness so that a full continuity transfer of load can happen between elements. Usually these joints require significant amounts of material and the presence of stiffeners, which make the detail more expensive (Bijlaard, 2006).

b. *Nominally Pinned Joints:* As told in 2.3.1) pinned joints should be able of *"...transmitting the internal forces, without developing significant moments..."* (Eurocode 3 1-8 Comitte, 2005). It is also important that the joint allows the expected rotation at the elements ends when design loads are applied.

c. *Semi-Rigid Joints:* The code classifies as semi-rigid joint to all those details that do not fit into any of the two previous categories. Furthermore, such joints are required to be capable of transmitting all internal stresses and forces.

The component method allows calculating the stiffness of the joint, in this way the result can be compared to the limits proposed by the code. In Figure 2-5, the boundaries for stiffness classification are presented:

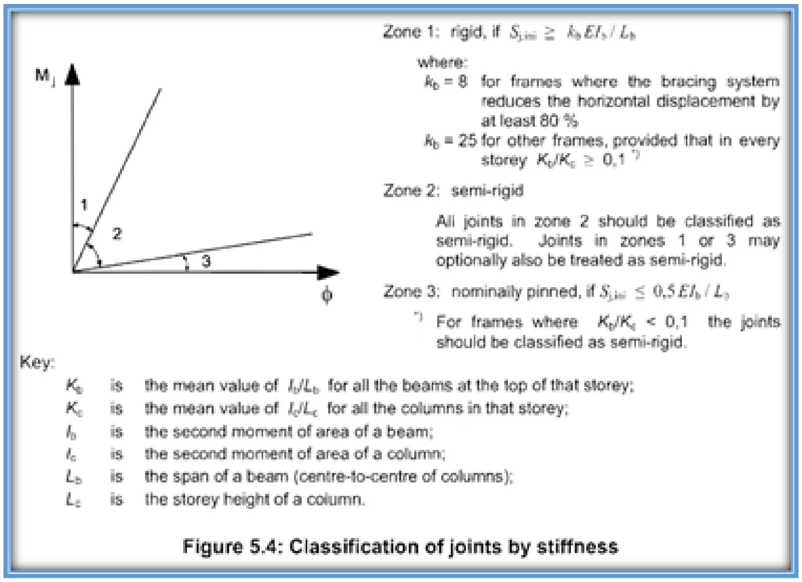


Figure 2 - 5: Joint Classification by Stiffness (Eurocode 3 1-8 Committee, 2005)

2.4.2) Classification by Strength

The classification by strength is of special importance when designing a structure in a seismic zone or under a plastic analysis. In these cases, it is wanted and required the development of a plastic hinge in the beam (in a column-beam joint). To ensure this, the joint must be able to withstand more than the plastic strength of the beam (EN1998 gives factors to take into consideration hardening of the material and other unforeseen parameters (see Annex A)).

The code classifies the steel joints in three groups by its resistance:

a) *Nominal Pinned Joints:* As in stiffness classification, this group of joints must be capable of transmitting internal forces without generating important bending moments. Moreover, the deformation capacity of these joints must be guaranteed under design loads. However, in this section the code adds a limitation, which considers a joint as simple (pinned) as long as its moment resistance is less than 25% of the moment resistance of a full strength joint. This resistance is compared with the plastic resistance of the elements that is joining.

b) *Full Strength Joints:* In order for a joint to be classified under this category, its resistance must be greater than the resistance of the elements it connects. In addition, the code gives certain parameters that these connections must meet. These parameters are presented in Figure 2-6.

c) *Partial Strength Joints:* all the joints that do not fulfill the requirements for the two previous categories are considered partial strength joints.

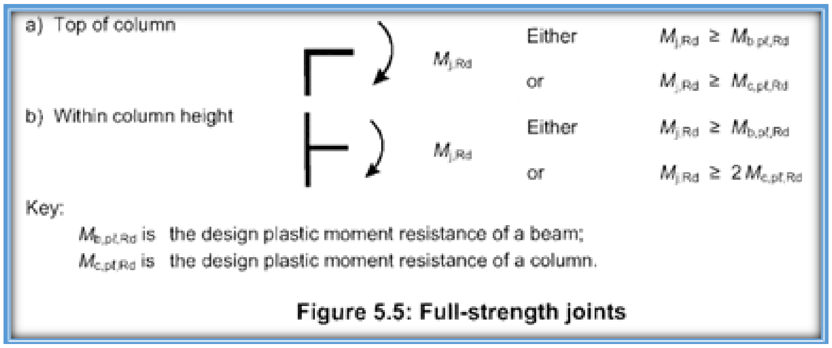


Figure 2 - 6: Full Strength joints Parameters (Eurocode 3 1-8 Committee, 2005

2.5 State of Art in Steel Joints Analysis

Present efforts are aimed to expand and improve the component method as a basis in the analysis of steel joints. Although the method is currently the most accepted for the design of joints, it has certain limitations and parameters that need to be improved.

The first major limitation in the use of the component method is its range of validity. The basic components (see Annex E) have been developed specifically for open profile sections type “I” or “H”. The calculation of the resistance of joints for tubular profiles follows the same idea as the design of simple joints. Here the different failure modes are analyzed and the one wit the lowest resistance will determine the resistance of the joint itself.

Moreover, not only the validity range of the method should be expanded. Some parameters of the existing basic components require further analysis. For example, when analyzing a joint where there is more than one element to connect (two beams to a column), it is necessary to apply the factor β, which depends on the moments and shear forces acting on the panel zone of the column web. The code gives values for the β factor, but studies have shown that a higher range is required for such values (Bayo, Cabrero and Gil, 2006). The importance of this factor can be observed when the drift of a frame is increased by the shear deformation of the column web be panel. This can lead to negative serviceability effects.

Another topic of analysis is the robustness of steel joints. (Simões da Silva, 2008) states *"current safety concerns for steel structures require that steel joints are designed to perform adequately under the wider range of loading conditions"*. This means that the current standards should be improved so that the designed joints can withstand beyond the normal static design load.

One more focus of concern is the limit of 10% axial force present in a beam, compared to its plastic axial resistance when its joint is designed. *"There is no background to justify this empirical limit of 10%"* (Simões da Silva et al., 2002). At the University of Coimbra, an experimental study was carried out, where different types of configurations for joints were tested under a range of axial forces applied. The study *"led to the introduction of a strict restriction that the axial force is smaller than 5% of the beam plastic axial resistance to be able to neglect the effect of the axial force on the connection"* (Simões da Silva, 2008). In this way, if there is a greater axial load in the beam, a more detailed analysis should be carried out where the interaction between moment and axial force is taken into account.

Other problems remain to be analyzed, for example, the validation of the design standards to large-scale joints presented in steel bridges, or the development of design procedures for joints subjected to seismic action, among others.

2.6 Numerical Methods for Joint Design

The increasing complexity of structures, the use of non-standard geometries and elements, the combination of different loads applied, among others parameters; demands the use of more friendly tools for the structural designers. Finite Element Analysis (FEA) is able to solve these kind of problems which can take important amount of time by solving them in an analytical formulation (if not impossible).

FEA is widely accepted and used in all engineering branches. It can be used in conjunction, as an alternative or as an expansion for experimental researches. Using Finite Element Method (FEM) instead of building it (structures, products, prototypes), is a cost effective solution in the experimental and structural practice environments. In this section brief explanation of FEA is made. In addition, the analysis of the method in the calculation of steel joints is addressed.

2.6.1) FEA

Finite Element Analysis (FEA) is a numerical tool, which is able to solve partial differential equations presented in the mechanical description of the behavior of a solid body. It also can be expanded to numerous types of problems like fluids and pressure problems. *"The technique is based on the premise that an approximate solution to any complex engineering problem can be reached by subdividing the structure / component into smaller more manageable (finite) elements"* (Mago and Hicks 2012).

Normally the FEA can be divided into three steps that most of the software implement in their interface.

1) Pre-processing: here the user must specify all the input necessary for the analysis. There are six principal parameters that need to be defined:
    a. *Type of analysis:* static, dynamic, transient, buckling, etc.
    b. *Geometry:* the geometry of the mathematical idealization of the structure is imputed by connecting different elements. Its nodes define the geometry. The use of symmetry is recommended at this stage to reduce the computational costs of the analysis.
    c. *Finite Element:* to choose the correct finite element is important for the accuracy of the expected result. For example, quadratic elements are better to reproduce bending deformation than linear elements, but they are considerably more expensive in computational costs.
    d. *Material Properties:* These properties are known or obtained by experiments. The material model can determine the level of analysis (linear or nonlinear). Nonlinearity due to material properties is known as physical nonlinearity. When big displacements are evaluated and the analysis takes into consideration changes in the trajectory of the load due to the deformation, geometrical non-linearity is applied.
    e. *Boundary Conditions:* are prescribed forces or displacements in the model. The behavior of the structure depends highly on them. The correct use of them is an important task in any analysis.
    f. *Time functions:* the way that the load are applied are prescribed in this parameter.

2) Processing: at this stage the differential equations are solved. In the displacement-based version of the finite element method, the equilibrium is solved through virtual work equations. Here the

program chooses a nodal displacement, with the displacement the strains can be calculated. After that and using the material properties (constitutive equations), the stress can be calculated. From the stresses the forces are estimated for each node. If the node reaches equilibrium then it is ready for the next load increment. If not, a new nodal displacement needs to be evaluated (interactions).

3) Post-processing: in this stage the results are obtained normally in terms of graphical deformed shapes, stress distribution, strain distribution, etc.

The benefit of using FEA is the possibility to simulate the behavior of any structure before it is manufactured. *“Once a model has been developed the analysis helps in evaluating the feasibility of the new design as well as trouble shooting failed designs already in the market and finding solutions without the need to prototype and waste time and money”*(Mago and Hicks 2012)

Nevertheless, it is important to remember that numerical analysis, in especial FEA, are able to obtain approximate solutions to problems. There is always error in the results and its magnitude depends on the different factors, in special in the size and type of the finite element. For this reason, it is of major importance the validation of models and software that use these tools.

2.6.2) FEA in Steel Joints

FEA can be used to solve a huge range of engineering problems; from which steel joints is not an exception. Krishnamuriny was the pioneer in the field of 3D Modelling of joints. For this, he applied shell elements into the analysis. It was found that shell elements cannot simulate in accurate manner the evolution of internal forces due to prying forces (equilibrium). From this point, an important number of alternatives and properties were added to a steel joint model. Combined nonlinear phenomena like material and geometrical nonlinearities, friction, slippage, contact bolt-plate interaction and fracture have to be introduced in the model. These are the major tasks that the modeling of a steel joint needs to overcome. Nevertheless, it was “*demonstrated that certain detailed features such as thread interaction do not need to be modelled to ensure useful results”* (Williams et al. 2009).

Within this framework, it is important to develop new tools/software that can be able to facilitate the definition of all the parameters previously mentioned. At the same time, the geometry needs to be imputed in a fast and precise manner. A proper equilibrium between analysis complexity and friendly interface needs to be found, in order to provide the structural engineers a fast, accurate and cost effective tool for the proper analysis of steel joints. The commercial finite element packages that are now in the market can overcome these problems. Nevertheless, these tools need to be validated and the user needs to be sure that his calculations are going to be in the safe side.

2.7 Hybrid Welds

In principle, all welds used until now are hybrid welds. The general idea of a hybrid structure or a hybrid weld is that there is more than one material quality involved in the process. In the present code and the previous ones, the filler material has not been considered. It was always used as a premise that the filler material must be stronger or at least have the same strength as the parts to be joined. In this philosophy, always the properties of base metals would be critical in the weld strength calculation. Its strength would be underestimated and therefore the design will be on the safe side. However, the new European steel joint code (2020) already takes into account the properties of the filler material. In addition, the code allows conditions of match, overmatch and now under match in welded connections. The use of High Strength Steel is also regulated in the new code, which opens up number of opportunities for structural designers.

Hybrid construction allows high strength steel to be used where it is most effective (trusses, beam flanges, etc), in places where the design of the element depends on the strength of the steel. And in places where the geometric properties (for example the web of a beam to obtain higher inertia) are critical in the design of the element, the use of mild steel is possible (which is cheaper). This same concept can be used in the local design of steel joints. This thesis will study the influence of having high resistance weld in a simple joint. In the event that it is feasible, a significant optimization in welding costs can be achieve. Thus, to realize a weld of 6 - 7 mm of thickness, three passes of welding are required. But if high strength weld is used and the weld thickness is reduced to 4mm, a single pass is necessary. In this way, welding time is saved. This is one of the most influential items within the production cost of a steel structure.

Hybrid welds follow the same system and design regulation, to which the fillet welds are subjected. The new code delivers standards and new values for the correlation factor required in the calibration of the regulation. These values were suggested after an intensive research program in Germany, where several parties collaborated to test the influence of the filler material on weld behavior. (H. P. Günther, J. Hildebrand, C. Rasche, C. Versch, I. Wudtke, U. Kuhlmann, M. Vormwald 2009).

This project will evaluate the overall behavior of high resistance weld. In other projects being developed at TU Delft, the local welding behavior will be evaluated using cross-shaped specimens. By subjecting these specimens to tension and fatigue loads, it will be possible to determine the actual strength of the high strength weld and the influence of the base materials when these are of different qualities.

Chapter 3 - Experiment

3.1 Introduction

Chapter 3th presents the design, production and results from a full-scale steel joint experiment. A simple joint (pinned) between a high strength steel (HSS) hollow section and a mild steel “I” shape beam will be studied. The aim of the experiment is to understand the behavior of high strength welds when they are used in hybrid construction. With the results, not only the calculation model and future formulation from the upcoming Eurocode 1993 1-8 (2020) will be tested, but also the understanding of the joint behavior will be improved. The influence of different quality steel materials of the base plates into the resistance of a HSS welds is another important goal in the experiment. The inside understanding of a steel joint (its rotation center), its idealization and assumptions are also going to be tested. In addition, this experiment will be used to set a benchmark in the comparison between the different methods to predict the capacity of a joint. Moreover, the accuracy of these methods will be tested. The experiment results will be analyzed to conclude the safety and accuracy of the finite element analysis and how it can be used in the daily practice of structural engineering.

Furthermore, a wide description of the experiment and the numerical predictions of it will be presented. A description of the procedures, material properties and boundary conditions will be detailed in order to allow future studies to replicate the model and use the experiment results for other investigation purposes.

The specimens were fabricated by ASK Romein and the test were performed at the steel laboratory (Steven Laboratory II) in the faculty of Civil Engineering of TU Delft.

Figure 3 - 1: Specimen

3.2 Objectives of the Experiment

The chosen joint for the experiment has many estimates and uncertainties. If the design of it is done using code rules, which is the normal practice in civil engineering industry, the designer needs to be aware of the simplifications and the validation range. The necessity of this knowledge is more evident when the steel joint is a non-standard detail since there is not a direct rule to consider the flexibility of the column face. The aim of the experiment is to evaluate these simplifications/uncertainties in order to understand how accurate and safe is to estimate a steel joint capacity using the code rules or a finite element analysis. In addition, normal practice techniques will be tested. In this way, the whole process of a steel joint production (from design, fabrication to real behavior) will be verified. The specific objectives of the experiment are:

a) To evaluate the performance of high strength welds in the assembly of hybrid steel structures (HHS column - mild steel beam and fin plate).

b) To estimate the real strength of welds when they are used in over-match and under-match conditions. It is going to be one of the first full-scale experiments where the filler material properties will be tested. Within this framework, the influence of different qualities of steel between the connected members and the welds will be studied.

c) To measure the real behavior of a simple joint through an unstiffened plate of a member (one side of a rectangular hollow section or the web of "I" shape section) to determine which one will be the correct way to idealize these kind of joints in a global analysis of the structure. For this, the real rotation center in the elastic and plastic zones will be calculated.

d) To compare the degree of accuracy and safety of the estimated values of the joint's capacity obtained through the use of Eurocode and the finite element analysis.

e) To test the proposed boundary conditions, which should resemble a perfect shear loading in a joint. This setup can be replicated to be used in future full-scale joints experiments.

f) To test the accuracy of IDEA StatiCa software in the design of no-standard joints that are common in industry practice.

3.3 Experiment Matrix

Three sets of specimens are planned for the experiment. Each set has three identical specimens in order to get a media of the results. The aim of the sets is to test the weld performance in three different situations: under-match, overmatch and ductility performance. The experiment matrix is presented in table 5-1.

Experiment Matrix

CODE	TEST		COLUMN	FIN PLATE	WELD		MEASUREMENT
	#	Code	Quality [MPa]	Quality [MPa]	Quality [MPa]	Thickness [mm]	
FIN_c690_w355_a3	1	1E1	690	355	355	3	Weld Strength (Undermatch)
	2	1E2	690	355	355	3	
	3	1E3	690	355	355	3	
FIN_c690_w690_a3	4	2E1	690	355	690	3	Weld Strength (Overmatch)
	5	2E2	690	355	690	3	
	6	2E3	690	355	690	3	
FIN_c355_w690_a5	7	3E1	355	355	690	5	Weld Ductility Behavior
	8	3E2	355	355	690	5	
	9	3E3	355	355	690	5	

Table 3- 1: Experiment Matrix

The aim of the first two sets is to determine the resistance (strength) of the welds by comparing the results from weld quality S355 and S690. In the third set, a full strength weld (according to the design rules) is projected. This will allow studying the behavior of the weld and how ductile it is. The weld specifications can be seen in Annex D. On the other hand, the calculation rules for full strength welds will be verified.

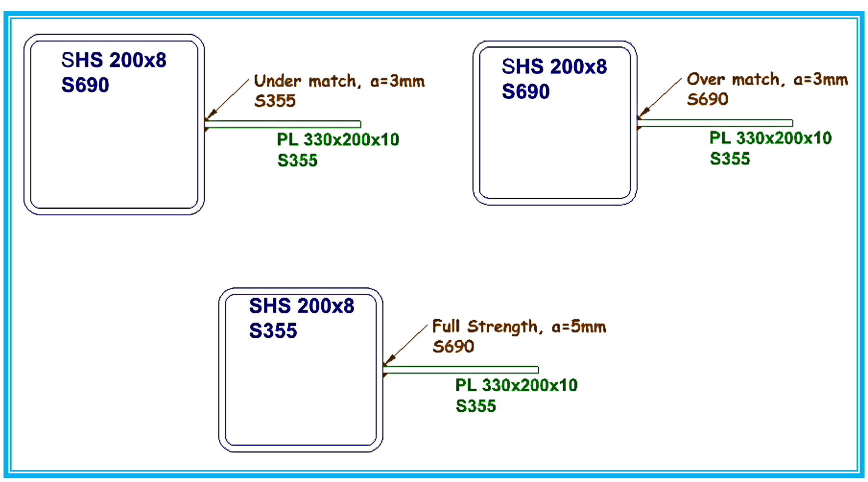


Figure 3 - 2: Experiment Matrix Figures

3.4 Drawings and Specifications of the Specimens

The size and properties of the profiles and plates were chosen in such a way that a real size joint can be tested. This type of joint is already used in the steel structures industry. The proposed joint can easily be found in a building or an industrial hall. With a real size experiment, the whole process of steel joint production can be verified from the design to production and from weld quality to real behavior of joint.

The specified properties are:

Element	Profiel/Plate	Length [mm]	Steel Grade	Number
Column	SHS 200x8	1050	S690	6
	SHS 200x8	1050	S355	3
Beam	IPE 400	600	S355	9
Fin Plate	PL 200x10	330	S355	9
Bolts	M 24	-	10.9	54

Table 3- 2: Elements Specifications

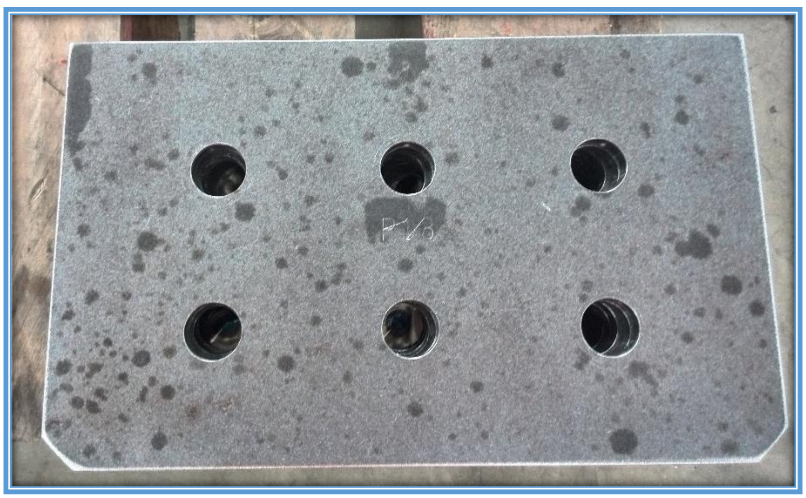
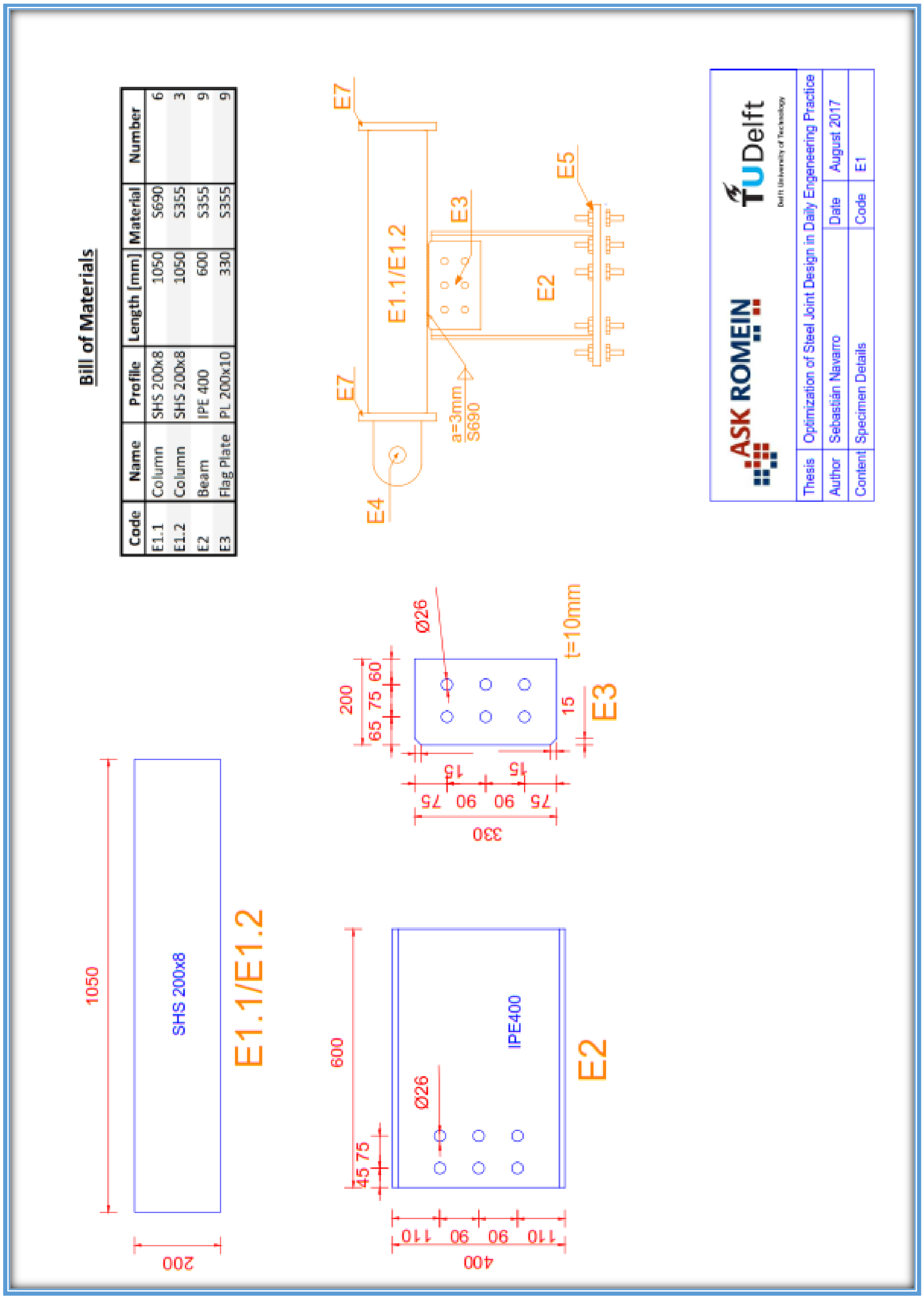


Figure 3 - 3: Fin Plate after Machining



Bill of Materials

Code	Name	Profile	Length [mm]	Material	Number
E1.1	Column	SHS 200x8	1050	S690	6
E1.2	Column	SHS 200x8	1050	S355	3
E2	Beam	IPE 400	600	S355	9
E3	Flag Plate	PL 200x10	330	S355	9

Figure 3 - 4: Workshop Drawings for the Specimens

3.5 Numerical Model

Experiments are important when validating equations, theories and code rules. They are also used to generate semi-empirical equations or charts to facilitate the design process. However, experiments are expensive and take a significant amount of time. Finite Element Analysis (FEA) can overcome these problems and through parametric studies, produce the same information that experiments do. Nevertheless, in order to use a finite element model, the validation of it should be the first step. The validation has to be done through experiments. The procedure is to validate a finite element model through a set of experiments and from this point, use the model as benchmark. Several new finite models can be produced based on the benchmark model.

The aim of this section is to give the necessary information for future replication of the finite model used in this thesis project and extended the use of it to the of study new parametric conditions.

3.5.1) Description of the Finite Model

Due to the unsymmetrical setup of the joint (see figure 3.4), symmetry simplifications were not possible to be done for modeling the experiment joint. Therefore, the whole joint was modeled. The finite element model was created using the software ABAQUS (Dassault Systemes Simulia Corp., 2012).

Geometry.-

Following the geometrical characteristics from the specimen specifications a set of solid elements (3D elements) was created (Figure 3 - 5: Bolt and Weld Geometry). The nominal geometry of the standard profiles was modeled, except for the bolts and welds. For the welds, a triangular cross section that accomplishes the specified throat dimension was used. For the bolts, a simplified tubular shape with circular ends was sketched. The threaded part of the bolt was not considered neither were the hexagonal shape of the nut and bolt head. The simplified shape of the bolt has a high accuracy of results (Kim, Yoon, & Kang, 2006) and saves important computational time due to the complex contact iteration between the nut and the thread shank. Furthermore, the bolts are not the critical failure elements for this experiment; thus, the results of the whole joint will not be significantly affected by the simplification.

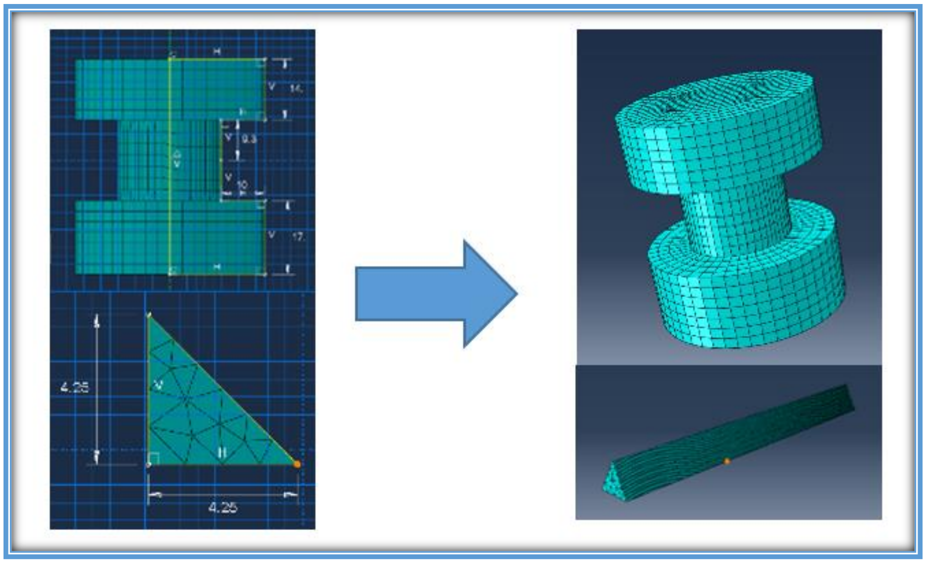


Figure 3 - 5: Bolt and Weld Geometry

Bolt Dimensions: 24mm shank diameter, 44mm nut and head diameter, 14mm height of the head and 17mm the height of the nut. For the weld, an isosceles rectangular triangle with the height equal to the specified weld throat was used.

Boundary Conditions.-

Especial boundary conditions were used for the experiment. In normal practice, the beam would be model with a free. The column extremes would be clamped and the load applied in the beam. Under this set of boundary condition, the beam end will rotate and the bending moments transferred along the beam will be cero (no bending deformation of the beam). In consequence, the beam will remain straight and the fin plate will rotate following the beam without generation bending stress (see figure 3-6). No hinge/spring is developed between the fin plate and the beam web, which is the assumption made by the code and the guide lines. The lever arm assume in the code is the distance between the weld (at the column face) and the center of gravity of the bolt group. Nonetheless, figure 3–6 shows a joint behavior were the lever arm is the distance from the column face and the end of the beam (were the load is applied).

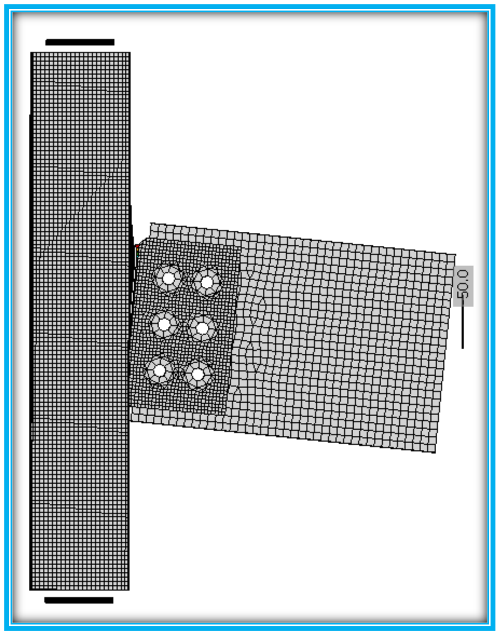


Figure 3 - 6: Common Simple Joint Analysis

However, this situation is not realistic because the beam cannot freely rotate at its end. The beam ill be "clamped" by its continuation (it is the right side if figure 3-6 is use as reference). In addition, the assumption mode by the code and the guide lines take into consideration the rotation between the fin plate and the beam web, by assuming a hinge in the center of gravity of the bolt group. In this frame of thinking, a new set of boundary conditions is proposed. The beam will be clamped at its end to simulate the real condition of the beam in a common structure. The load will be applied in one of the column extremes and both of them will be restrain to move in the out-of-plane and horizontal direction. In this way, a pure axial load, that simulates a "pure" shear transfer of load, can be simulated and the Eurocode assumptions for simple joints can be replicated. Figure 3- shows the behavior of the joint under the proposed set of boundary conditions.

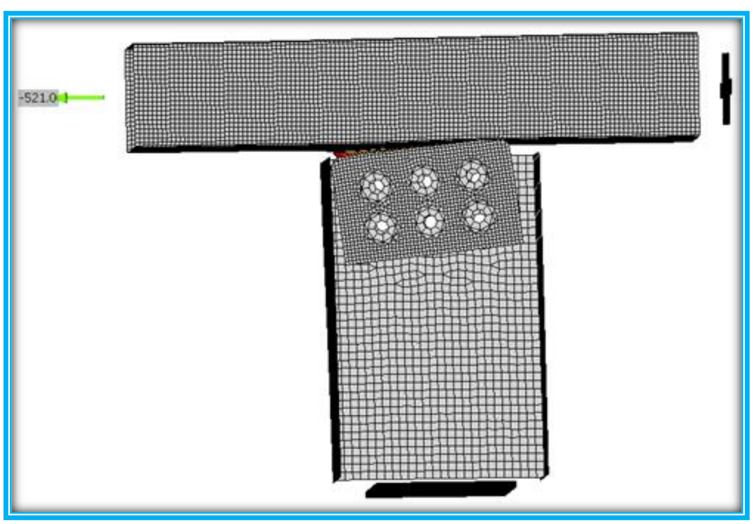


Figure 3 - 7: Proposed Analysis of the Simple Joint

It is possible to see in figure 3–7 the development of a rotation center in the fin plate and at the column face. The rotation between the beam web and the fin plate is proper simulated and a joint analysis closer to reality is obtained. In figure 3–8 the idealization of both set of boundary conditions can be seen.

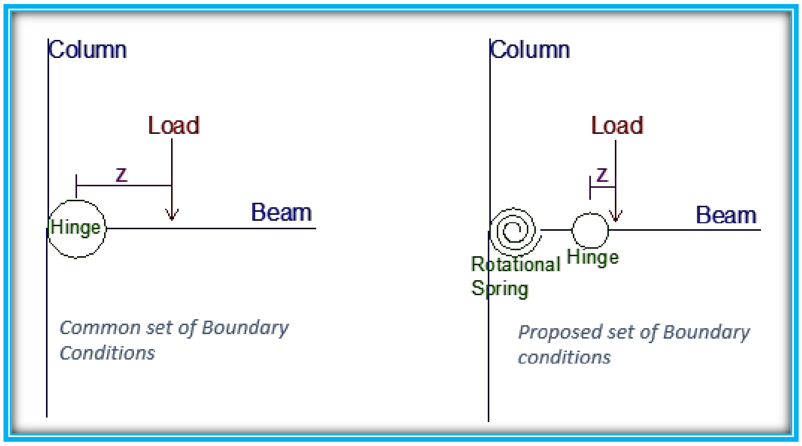


Figure 3 - 8: Common Boundary Conditions vs Proposed Boundary Conditions

The proposed boundary conditions were replicated in the experiment and in the ABAQUS numerical model. The boundary conditions where specified in the real locations where the experimental set-up was able to reproduce those conditions. With these conditions, a shear transfer of the load is achieved avoiding possible distortions due to the eccentricity between the fin plate and the beam. These eccentricities will introduce second order effects leading to second order moments, which are not taken into consideration for this kind of detail when they are used in practice. The column was pulled from one of it ends to transfer a tension load. In this way, local buckling due to the load insertion is prevented.

The boundary conditions were applied at a master node (reference point), which at the same time has its degrees of freedom coupled to all the nodes from the surface of the cross-section at the specified position in the elements. This not only allows the easy control of the applied properties at the boundary conditions, but also the extraction of the support reactions is simpler.

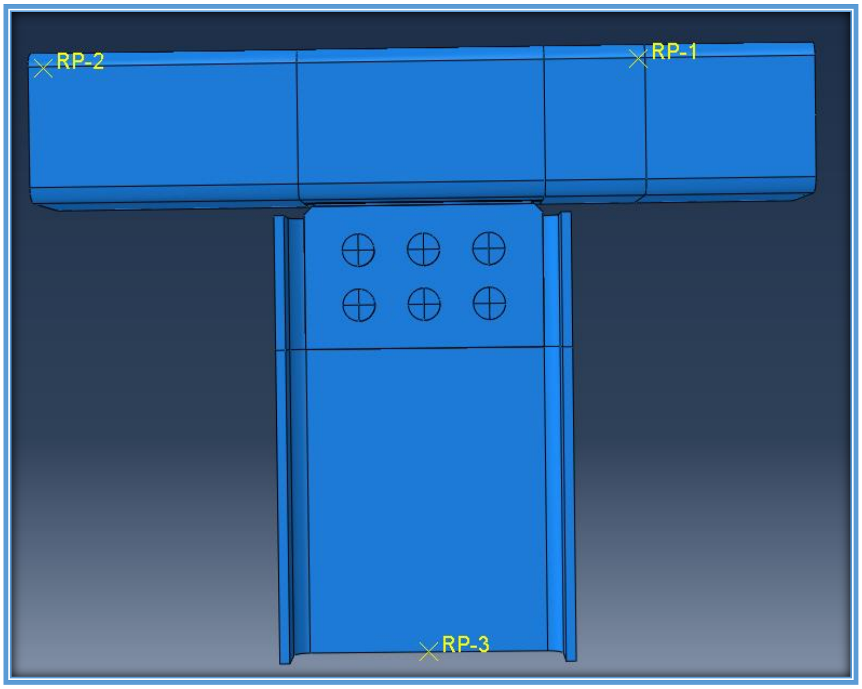


Figure 3 - 9: Location of the Master Nodes at the 3D Model

Contact.-

Surface to surface contact and friction are important interactions that have to be used in a steel joint model. For the bolts-fin plate and the bolts-beam web contact properties, friction and hard contact were used. The friction was applied in the contact surfaces between the nut and the bolt head to the fin plate and the beam web, with a friction coefficient (μ) of 0.4 (this property will be activated only when high deformations occur in the joint). At the same time, hard contact between the shank and the inside surface of the hole, was specified. This property will allow the study of the bearing behavior of the bolts and the plates. Pre-tension of bolts was not considered for the model.

For the welds, a tie constrain was applied between the weld surface, the column face and the fin plate face. A small gap between the fin plate and the column face was specified (see Figure 3 - 10: Gap between Column Face and Fin Plate). The aim of the gap is to avoid errors in the constraints. If the two elements are in contact in the common corner of the weld surface, column face nodes and fin plate nodes will be directly coupled, producing a direct transfer of the load between these two elements and the weld will remain unstressed.

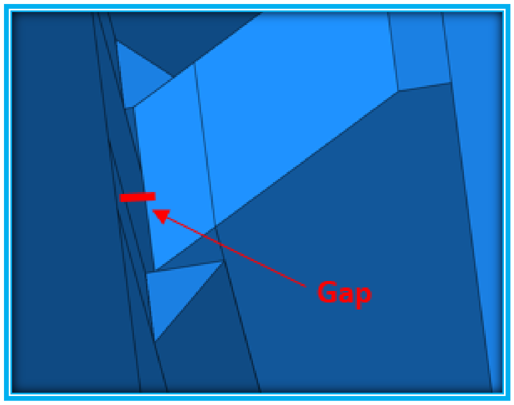


Figure 3 - 10: Gap between Column Face and Fin Plate

3.5.2) Material Properties

Two different material models were used:

1) Perfect plastic model, to compare the results between the two finite element software and the rules from Eurocode.
2) Nominal Hardening model that is specified in Annex C of EN19931-5, to take into account hardening of the steel. The hardening slope is the reduced stiffness (E/100) proposed in the code and it was assumed to be isotropic. The results from this model were compared to the experiment behavior results (see 3.9).

For all the elements, the theory to calculate the yielding stress was based on the Von Mises Plasticity Theory. An elastic modulus (E) of 210000 [N/mm^2], a density of 7.85e-9 [ton/mm^3] and a Poisson ratio (v) of 0.3, were specified.

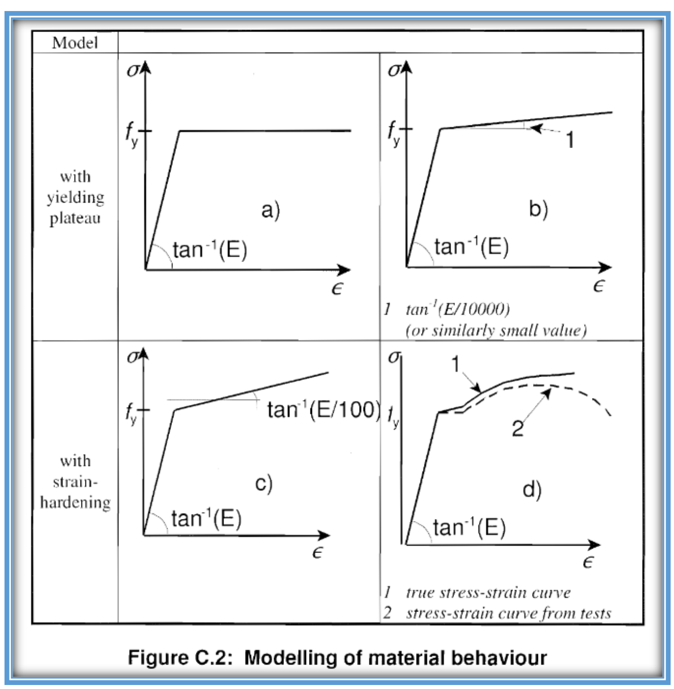


Figure 3 - 11: Material Models (EN1933 1-5 Annex C)

3.5.3) Type of Finite Element

For the profiles and plates, hexagonal 3D (bricks) finite elements were used. However, for specific parts of the elements where the geometry is complex and the hexagonal elements were too distorted, pyramidal elements (wedge) were used. This is the case of the rounded part of the IPE profile that gives bad quality hexagonal elements. The bricks elements are preferred over the triangular since they are more accurate in the stress/strain measure and distribution. The wedge elements have a stiff bending behavior (Dassault Systemes Simulia Corp., 2012). It is recommended to use the wedge elements far from the zones where accurate results are needed. Nevertheless, it is better to use these elements than using high distorted hexagonal finite elements.

For volumetric elements, a minimum of four finite elements through the thickness should be used to be able to capture the actual stress distribution. This parameter is more important when the bending stress is applied perpendicular to the thickness of the section.

For the hexagonal finite elements, a cubic 8 nodes (C3D8R) element was selected. This linear element has six degree of freedom in its 8 nodes and uses reduced integration for its analysis. In addition, it contains hourglass control. The element only has one integration point in the center unlike the (C3D8I) cubic element, which has four integration points. The difference in the number of integration points for an element has some consequences. The C3D8I, which uses full integration, is more accurate for bending stress at a very high computational cost. Additionally, this element does not suffer from shear locking because its analysis uses supplementary standard shape functions (bubble functions). The bubble functions have zero value at the nodes but non-zero in between them. In the joint of the experiment, bending is not dominant and the difference in the results obtained from the two elements (see Figure 3 - 12: Reduced vs Full Integration) is not significant. Moreover, the C3D8I analysis was four times slower for this specific case. Consequently, the reduced integration element was chosen. For the wedge element, a linear six node element with reduced integration (C3D6R) was used.

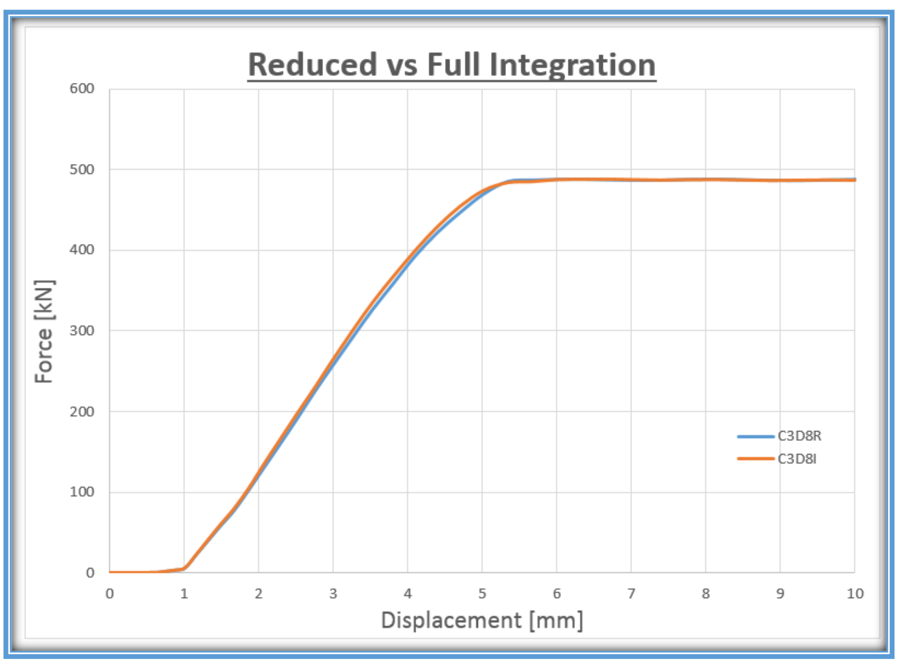


Figure 3 - 12: Reduced vs Full Integration

It is important to mention that a quadratic element can probably give more accurate results. However, in the ABAQUS/Explicit package (see 3.5.4), which is the solver used, only linear finite elements are available (Dassault Systemes Simulia Corp., 2012).

C3D8R finite element has only one integration point in the middle. In order to reduce the error from the numerical analysis, small elements (dense mesh) are needed to capture a stress concentration at the boundaries of structure elements. An approximately number of 170000 elements where used in the model. High dense mesh zones were used in the connection between the column face, welds and fin-plate. In the extreme part of the members, a less dense mesh was specified to decrease the computational time.

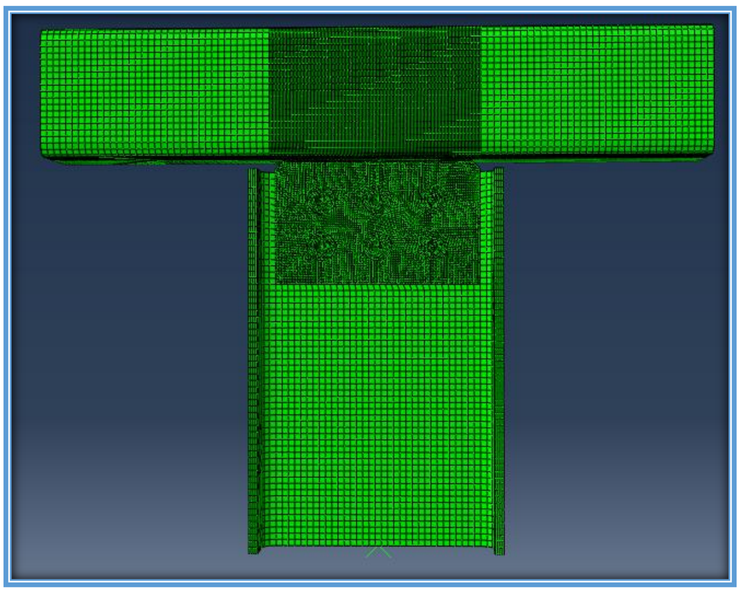


Figure 3 - 13: Mesh Density in the 3D Model

3.5.4) Solver – Computational Procedure

ABAQUS/Explicit package was chosen as the solver for the analysis of the experiment joint. This solver has the benefit of not suffering from converge problems unlike a static or an implicit solver. Moreover, due to the hole clearance that was taken into consideration in the model, an static or implicit analysis cannot be performed. This is caused by the fact that at the initial stage, the bolt needs to move from the center of the hole to the surface of the hole until the contact between the shank and the hole surface is achieved. In this first stage, the stiffness of the joint is zero and the load flow has a gap.

ABAQUS/Explicit can solve complex contact problems like the one present in the experiment joint. This is possible because implicit and static solvers must do iterations to determine the solution for a non-linear problem. On the other hand, explicit solver determines the solution by "explicitly" finding the new kinematic state from the end of the previous increment. The explicit dynamics is a mathematical technique for integrating the general equations of motion through time. This technique is also called "Forward Euler or Central Difference Algorithm".

Nevertheless, the solver has a draw back. It can be called "Inertia locking". If the analysis is too fast, the inertia of the mass of the elements will lead a different load and stresses flow than the one expected from a quasi-static analysis. This is because the wave propagation is affected by the mass inertia. Inertia locking can easily be checked by equilibrium of energies in the system. A quasi-static analysis is desired for the experiment model. For this purpose, the introduction of the load and the speed of the analysis should be slow enough to overcome the mass inertia lock. A practical example to understand this phenomenon can be illustrated with a beer can and a bullet. If somebody presses a bullet against the face of the aluminum can, large deformations may occur across the face of the can. If the pressure applied is too high, the shape of the bullet can be printed in the face of the can. Now, if another beer can is fastened in the extremes and the bullet is shot, a perfect circular hole will be produced (in reality two holes, in and out holes). In this occasion, the load was applied at such a high rate that the mass inertia does not allow the stresses and strains to propagate all over the can. High damage happened only in the zone really close to the bullet hole. It can be said that the particles of the can that are far from the applied load point, will not "feel" any stress from the bullet.

The idea of the example can be applied to the experiment joint. If the tension load applied in the extreme of the hollow section is too fast, yielding of the cross section close to the boundary will occur. The load will not be transferred to the joint and the beam support reactions will be zero. In order to achieve a quasi-static analysis with the explicit solver, a smooth loading curve should be specified. Sudden changes in the applied load rate can lead to the production of stress waves, which introduce noisy and inaccurate results. For this reason, an artificial increment of the load rate, 1/1000, was used in the step load. In addition, the mass scaling factor was small (0.005) and was applied at every increment of the load through the step. Scaling mass reduction or increase of the step time have the same effect in the analysis. The difference is that it is necessary to apply a mass scaling factor of 10^2 to have the same effect of 10 factor in the increment of the loading rate.

In fact "the goal is to model the process in the shortest time period in which inertial forces remain insignificant" (Dassault Systemes Simulia Corp., 2012). In other words, we want to perform a dynamic analysis, in which only the steady state solution is presented in the process.

To determine a suitable applied load rate, a frequency analysis should be executed. With the frequency analysis, the fundamental frequency can be computed, which will be used to calculate the speed of wave propagation through the model. A recommended load rate is the wave speed divided by a factor of 50 (Dassault Systemes Simulia Corp., 2012). The next step is to perform the quasi-static analysis. To check if the applied time is correct and that the analysis do not suffer from "inertia locking" and the results are accurate enough, some parameters should be checked:

1. Equilibrium in the structure. The difference between the applied load and the reaction forces should be very small. Less than 1%. Note that due to the nature of the solver, having the exact value for the applied forces and the reaction forces is extremely hard and computationally expensive to achieve.

2. The load curve and the reaction curve (force over step time) should not be noisy. This means that the curve should be smooth and a "wave" behavior should be avoided.

3. The Kinematic Energy should be small compared to the internal strain energy. A recommended value is between 5% to 10% (Dassault Systemes Simulia Corp., 2012). For the experiment numerical model, less than 1% was achieved.

4. The variation of the Kinematic Energy through time needs to avoid a noisy behavior.

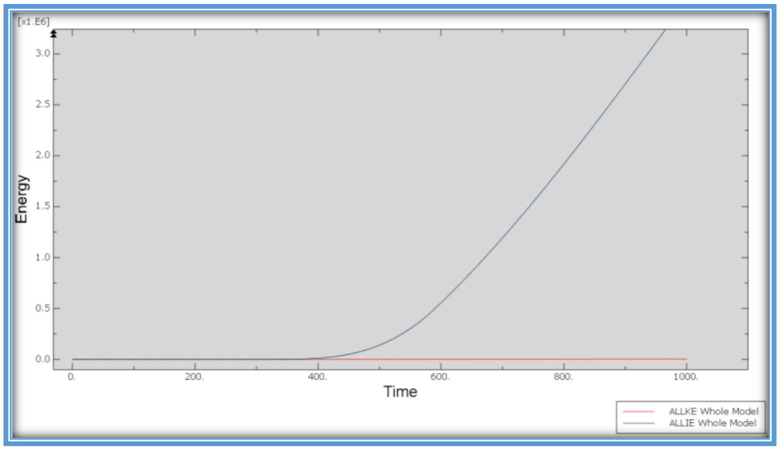


Figure 3 - 14: Internal vs Kinematic Energy in the COL690WELD355 Model

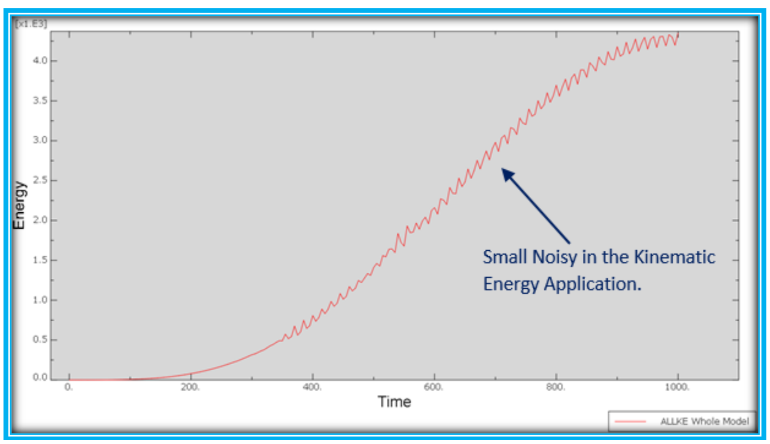


Figure 3 - 15: Kinematic Energy Model COL690WELD355

3.6 Laboratory Set Up

The step previous the experiment execution is the design and specification of the laboratory setup. This includes the connections between the specimen and the frame. In order to ensure that the applied load and the stiffness necessary to produce failure in the joint is achieved, numerical and hand calculations were performed. The analysis needs to consider the available elements (Jack, bolts, beams, etc.) in the laboratory. Rules like the distance from hole to hole and maximum jack elongation were also considered.

3.6.1) Joint Resistance Estimation

The results of the analysis are present below:

Analysis	Material Model	Limit	Shear Load [kN]
Hand Calculation	Eurocode Rules (No Safety Factors)	Weld failure	536.95
IDEA StatiCa	Perfect Plastic Model	5% Plastic Strain	505
ABAQUS	Nominal Hardening	Weld Failure	686.9

Table 3- 3: Experiment Joint Estimations

A maximum of 687 [kN] is predicted. In order to ensure the small deformation of the frame and to take into consideration the over-strength due to real material properties and geometry, a load of 800kN is chosen to analyze and design the experiment setup. This load will be used as services level load, this means that plasticization of steel in the frame and connections will be controlled to be as small as possible.

In addition, to simulate the selected boundary conditions for the experiment, the lateral (out of plane) and vertical reactions are calculated to design the frame setup.

Reactions Forces		
Direction	Left Column Support [kN]	Rigth Column Support [kN]
Out of Plane	-1.75	0.82
Vertical	-52.5	61.8

Table 3- 4: Reactions at the Column Extremes

The reaction loads direction can be understand from Figure 3 - 9: Location of the Master Nodes at the 3D Model. The negative in the vertical direction means downwards and the negative in the out-of-plane direction means pointing to the viewer. It can be concluded that the joint is trying to rotate anticlockwise (from the top view). With this information, the frame supports and connections were designed.

3.6.2) Frame Analysis

Once the applied load (800kN) was chosen, beam type elements and a linear elastic analysis was performed on the frame. The software Autodesk Robot Structural Analysis Professional was used for this calculation. The self-weight and the applied load in the specimen were considered. Two different options of analysis were performed. The first option was a rigid idealization of the experiment joint and the second one was a hinge (pin) idealization of it.

After the first analysis, a middle support for the long beam was proposed since the deformation in the frame was significant. After that, the second analysis showed the following results:

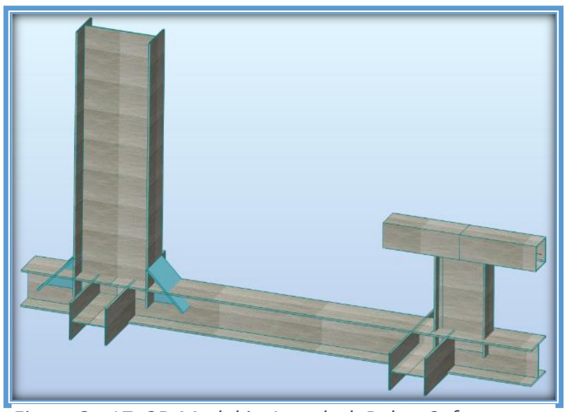


Figure 3 - 17: 3D Model in Autodesk Robot Software

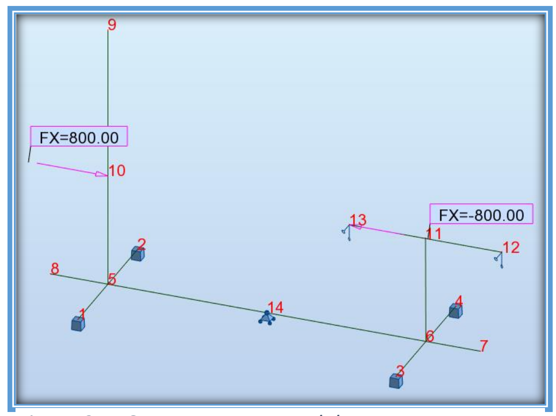


Figure 3 - 16: Beam Frame Model

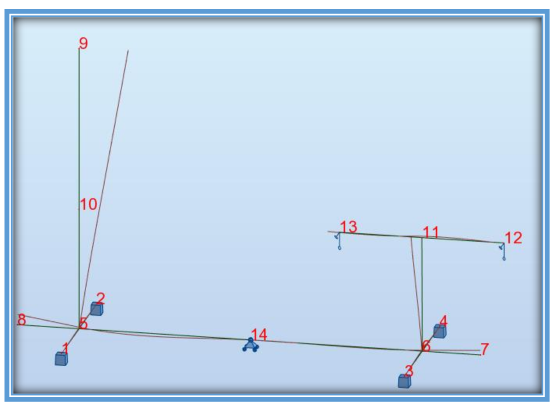


Figure 3 - 18: Rigid Model Deformation

Node/Model	Rigid Model		Pin Model	
	Ux [mm]	Uz [mm]	Ux [mm]	Uz [mm]
10	7.5	0.6	7.4	0.6
13	-4.4	0.0	-6.9	0.0

Table 3- 5: Frame Node Global Displacements

The node displacements gives the idea of how rigid is the frame and its results are satisfactory. The constraints for out-of-plane and vertical displacements of the column extremes were considered during the analysis.

3.6.3) Frame Details

By taking the real measures of the frame elements, the Jack, the supports and the brackets; the following setup detail for the experiment is proposed (see figure 3-14).

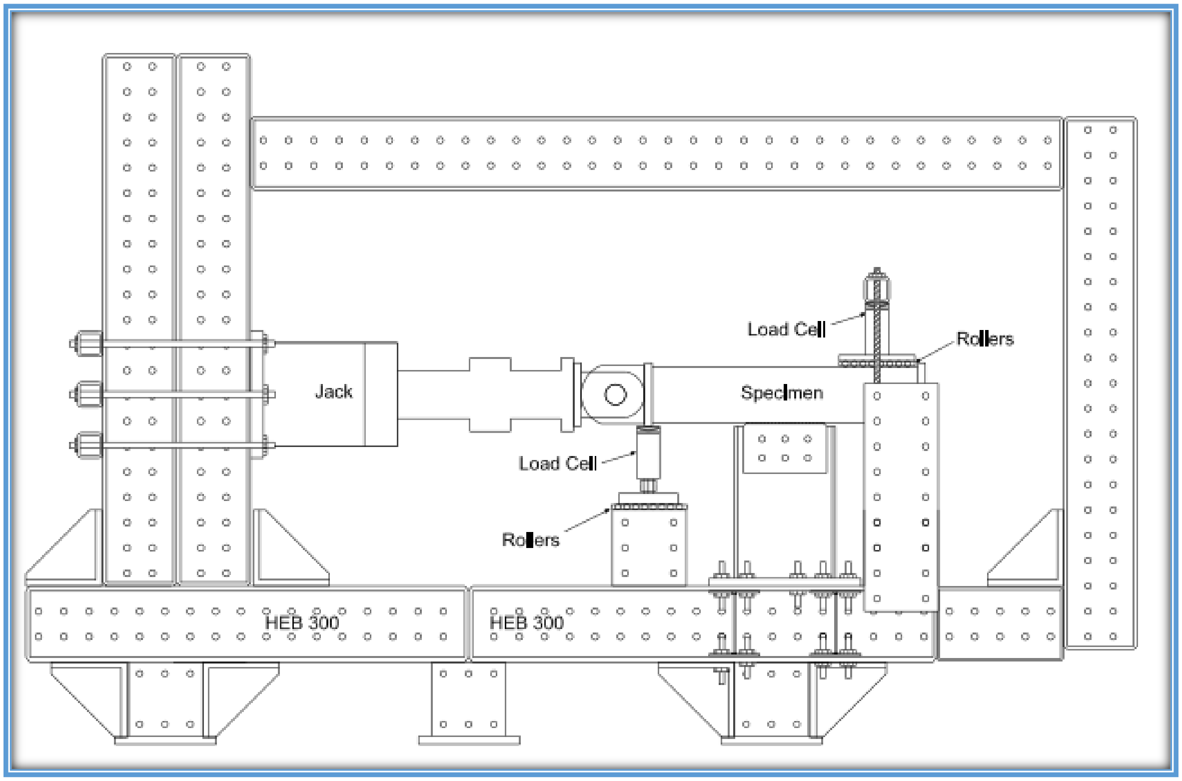


Figure 3 - 19: Frame Setup

3.6.4) Frame-Specimen Connections Design and Detailing

Part of the experiment setup is the design of the connections between the frame and the jack with the specimen. The parameters taken into consideration are:

- An applied load of 800kN in order to take into account the hardening of steel, the real strength of materials and other possible uncertainties in the experiment.

- The plasticization of the connections should be avoided, in order to no have a big influence of the deformation of the connections in the reading deformations of the experiment joint.

Three joints need to be designed:

1) *Beam Clamp to the Frame.-*
   The design of this "moment resisting joint" needs to follow the rules from the frame setup. One of these rules is to have holes every 100mm for M24 bolts. This detail needs to be a rigid joint, to resemble the real conditions of the joint in normal structures (chosen boundary conditions). The design showed that stiffeners should be used in order to avoid failure of the HEB300 beam from the frame.

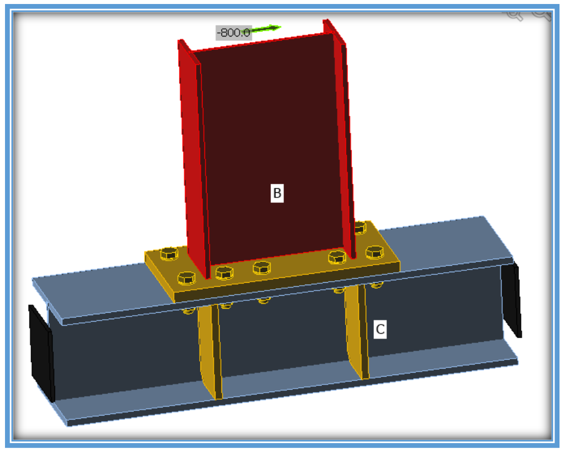


Figure 3 - 20: Beam Clamp Model in IDEA StatiCa

2) *Column Pin-Plate.-*
   The force was applied to the specimen through pulling one of the extremes of the column with a Jack. A tension load was chosen to avoid local and lateral buckling of the column. The design of the pin-plate geometry was executed following the recommendations in Eurocode and later analyzed in IDEA StatiCa to check that the plastic strains are low.

3) *Jack to Pin-Plate.-*
   This is the only detail that will be the same for the nine tests. Therefore, plastic strains are a critical factor and they should be avoided. Taking into consideration fabrication parameters, the same plate thickness as the column-jack connection was used for this connection.

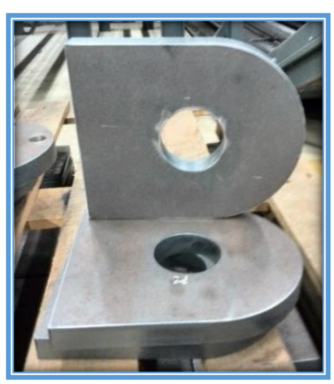


Figure 3 - 21: Pin Plate Production (E4)

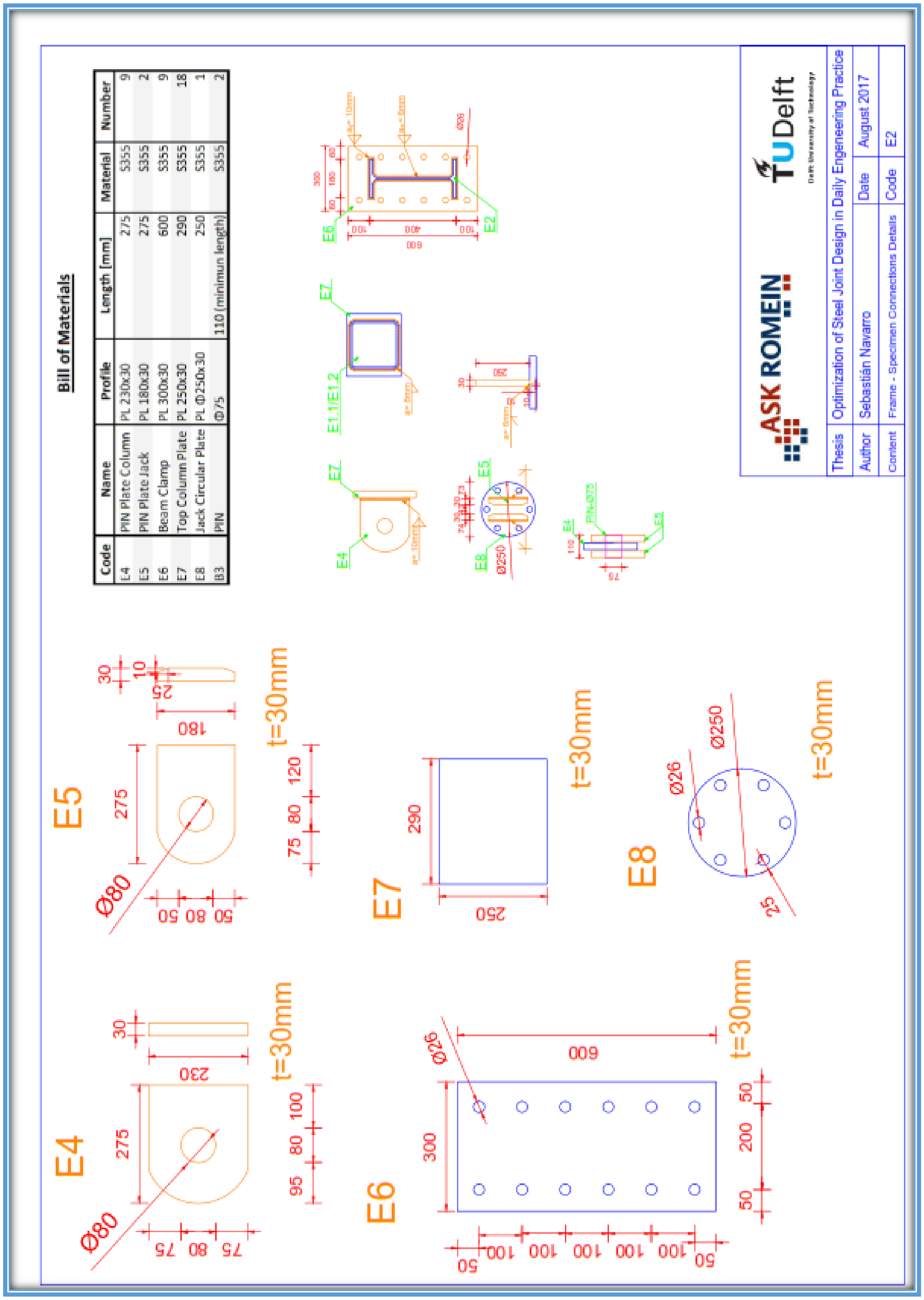


Figure 3 - 22: Workshop Drawings for the Connections to the Frame

3.6.4) Strain Gauges and Displacement Measurements Location

The aim of the strain gauges locations are to measure the bending stresses distributions in the beam´s cross section to give an insight of how the bending stresses are changing through the length of the beam. The experiment joint supposes to be a simple (pinned) joint. Nevertheless, one of the goals of the experiment is to measure its the real behavior. One of the parameters was tested is the real location of the rotation center. In this way, the rules assumption of the location of the rotation center can be validated.

The displacement measurements will help both to understand the behavior of the joint and to test the influence and performance of the boundary conditions and the frame.

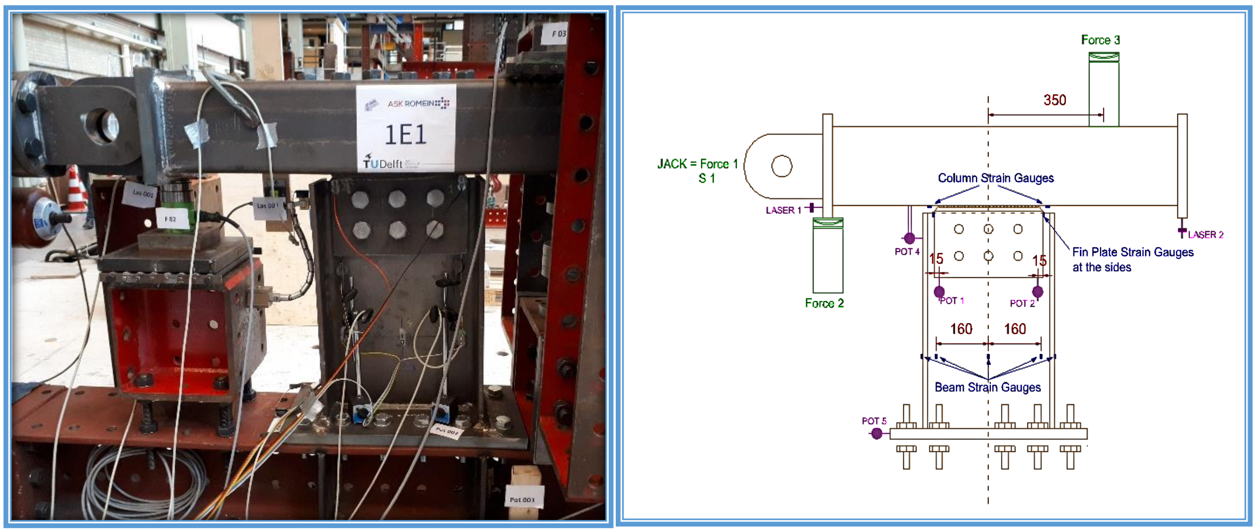


Figure 3 - 23: Instrumentation Location

3.7 Analytical and Numerical Predictions

3.7.1) Geometry Description

In previous points, the description of the numerical model and the geometry of the joint were given (see Figure 3 - 4: Workshop Drawings for the Specimens). In this section, a summary of the properties of the joint is tabulated. The geometry dimensions are the actual dimension measure from the specimen itself. However, for the material properties, nominal values from the code and the material specifications were used.

Element		Column	Beam	Fin Plate	Bolts	
Profile		HSS 200x8	IPE 400	PL330x200x10	M24 - 10.9	
Dimensions [mm]	*Length*	1050	599.00	-	24.00	Bolt diameter (tension) [mm]
	Heigth	203.00	402.00	230.00	21.30	Thickness of bolt nut [mm]
	Width	202.00	177.00	100.00	35.80	Across points dimension of bolt head & nut
	Flange Thickness	8.80	12.70	8.00	15.80	Thickness of bolt head [mm]
	Web Thicknes	8.40	8.65	-	4.00	Thickness of ALL washers [mm]
	Rolling Radius	20.00	21.00	-	3.00	Fillet Welds Throat
Material Properties [kN/mm²]	*Yielding Stress (fy)*	355.00	355.00	355.00	900.00	
	Ultimate Stress (fu)	510.00	510.00	510.00	1000.00	
	Young Modulus	210000.00	210000.00	210000.00	*210000.00	

Table 3- 6: Experiment Joint Material and Geometry Properties

3.7.2) Hand Calculation (EN 1993 1-8 (2005))

The design resistance of the joint calculated by using the Eurocode formulations presented in the guidelines (Jaspart, 2016) was of 365 [kN] (see Annex B). The purpose of the experiment is to test the behavior of the weld. With this objective, the design was performed in such way that failure can happen in the weld. Nevertheless, shear resistance of the fin plate and the shear out of the beam have resistance close to the failure of the weld.

For the weld resistance calculation, the concept of hybrid weld (see chapter 4) was used. In this case the filler material properties were used in conjunction with the new normative present in the future Eurocode for steel joints (EN1993 1-8 (2020)).

Failure Mode		EN 1993 1-8 (2005) [kN] With Safety Factors	Without Safety Factors
$V_{RD.1}$	Shear Resistance of Bolts	409.25	1059.00
$V_{RD.2}$	Bearing Resistance of the Fin Plate	594.62	1099.88
$V_{RD.3}$	Shear Resistance Fin Plate: Gross Section	532.57	614.88
$V_{RD.4}$	Shear Resistance Fin Plate: Net Section	593.61	742.01
$V_{RD.5}$	Shear Resistance Fin Plate: Shear Block	585.26	879.02
$V_{RD.6}$	Bending Fin Plate	628.61	628.61
$V_{RD.7}$	Buckling Fin Plate (Factor)	628.61	628.61
$V_{RD.8}$	Bearing Resistance of the Beam Web	446.70	684.86
$V_{RD.9}$	Shear Resistance Beam Web: Gross Section	938.18	939.18
$V_{RD.10}$	Shear Resistance Beam Web: Net Section	884.37	1150.58
$V_{RD.11}$	Shear Resistance Beam Web: Shear Block	461.63	540.07
$V_{RD.12}$	Shear Resistance Column Web	5048.17	6310.21
$V_{RD.13}$	Weld Resistance (EN1993 1-8 2020)	365.00	512.50

Table 3- 7: Experiment Joint Failure Modes - Shear Resistances

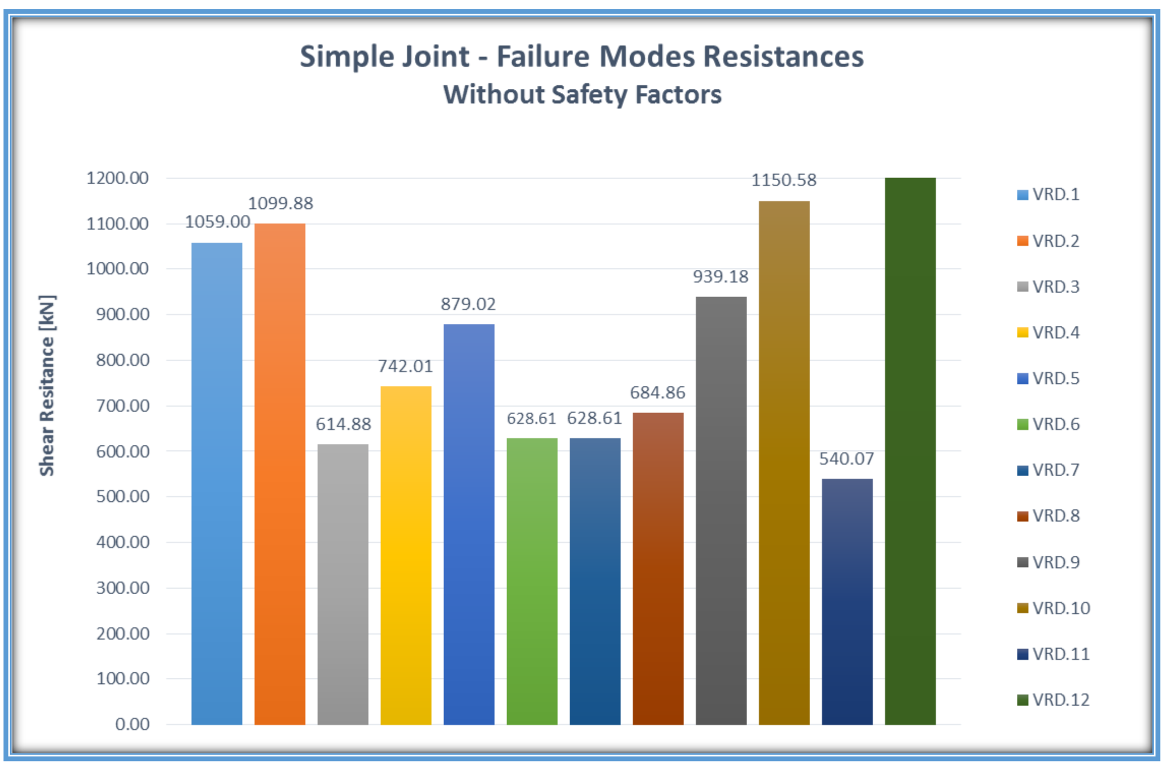
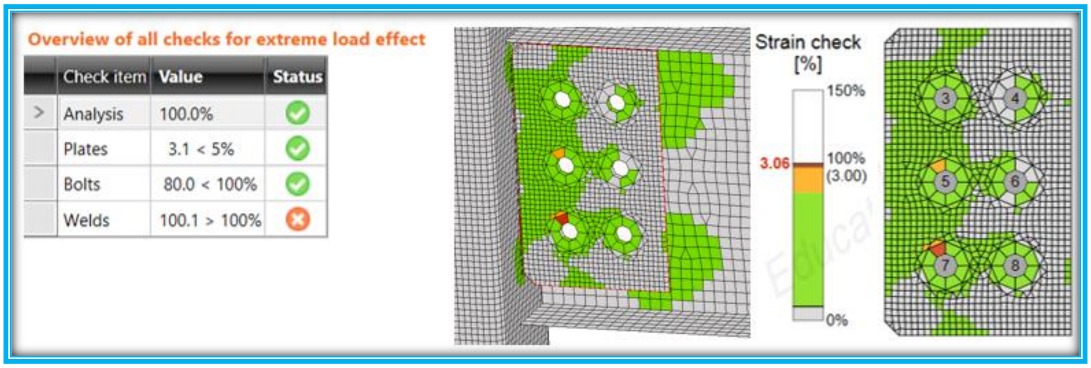


Figure 3 - 24: Experiment Joint - Hand Calculation Results

3.7.3) IDEA StatiCa Analysis

Using the software IDE StatiCa, a shear resistance of 521.0 [kN] is predicted for the joint. The failure happened at the weld (see Figure 3-22). Nonetheless, plastic strains are present in the Fin Plate, suggesting some bearing happening.


Overview of all checks for extreme load effect

Check item	Value	Status
Analysis	100.0%	✓
Plates	3.1 < 5%	✓
Bolts	80.0 < 100%	✓
Welds	100.1 > 100%	✗

Figure 3 - 25: Experiment Joint - Results and Plastic Strain Distribution (IDEA StatiCa)

3.7.4) ABAQUS Analysis

For the ABAQUS numerical analysis, two analyzes were performed. One using Ideal Plastic Material Model and by taking into consideration the 5% Plastic strain limitation to compare it with the hand calculations. The second analysis was performed using Nominal Harding Material Model, and it was compared to the actual behavior of the joint in the experiment.

The Ideal Plastic Material model gave a shear resistance of the joint of 551 [kN]. The failure mode that is limiting the joint resistance is plasticization of the weld. Nevertheless, Figure 3 - 26: Experiment Joint - Strain/Stress Distribution (ABAQUS)shows bearing failure in two bolts. With these results, the appearance of some failure modes when the joint fails is expected. Important deformation can be expected at the holes of the fin plate and the beam web at the same time big strains at the fin plate suggesting some shear deformation of the gross section.

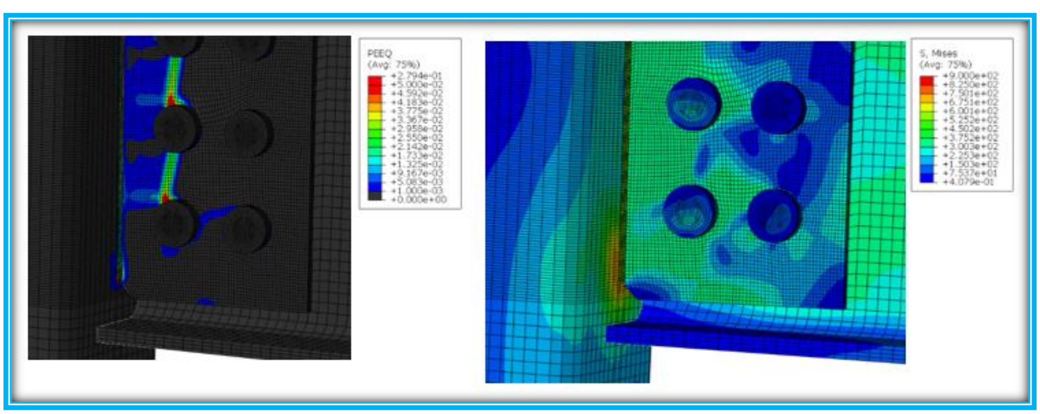


Figure 3 - 26: Experiment Joint - Strain/Stress Distribution (ABAQUS)

The Force vs Displacement curve presented in Figure 3 - 27: Experiment Joint - Force vs Displacement Curve (ABAQUS) shows the behavior of the joint. It can be seen that at the beginning no force is developed, this occurs due to the bolt hole clearance.

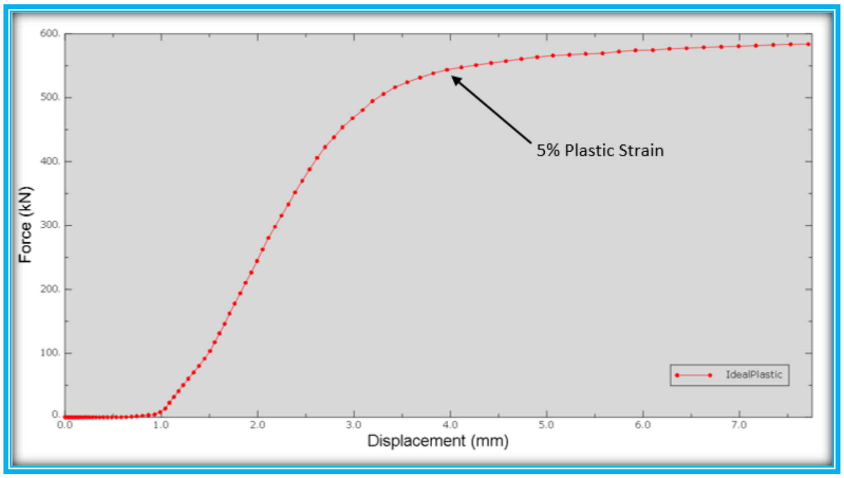


Figure 3 - 27: Experiment Joint - Force vs Displacement Curve (ABAQUS)

The Nominal Hardening Material Model gave a maximum shear resistance of 686.9 [kN]. The failure is still happening in the weld; nonetheless, big plastic strains are occurring in the first column of bolts. Big bearing deformation is still expected to happen in the experiment.

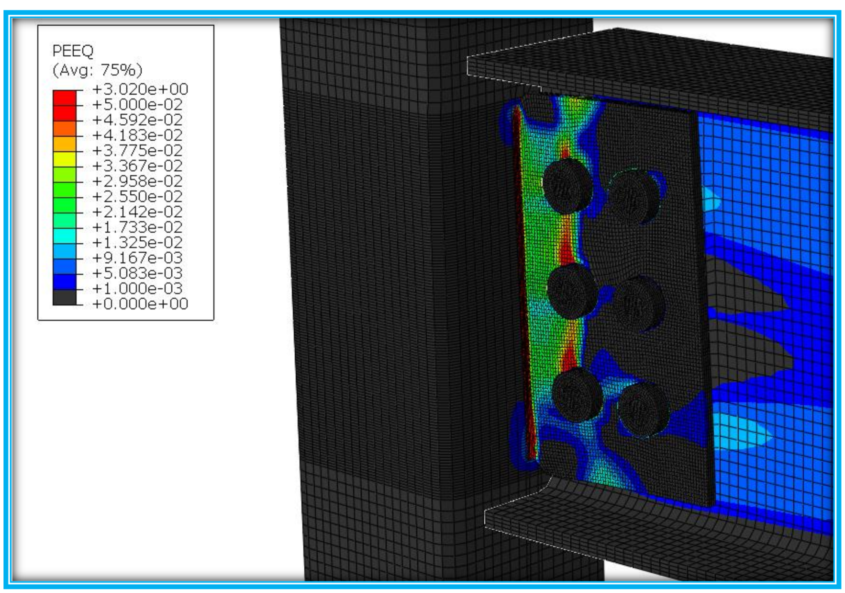


Figure 3 - 28: Experiment Joint - Plastic Strain Distribution (Nominal Hardening)

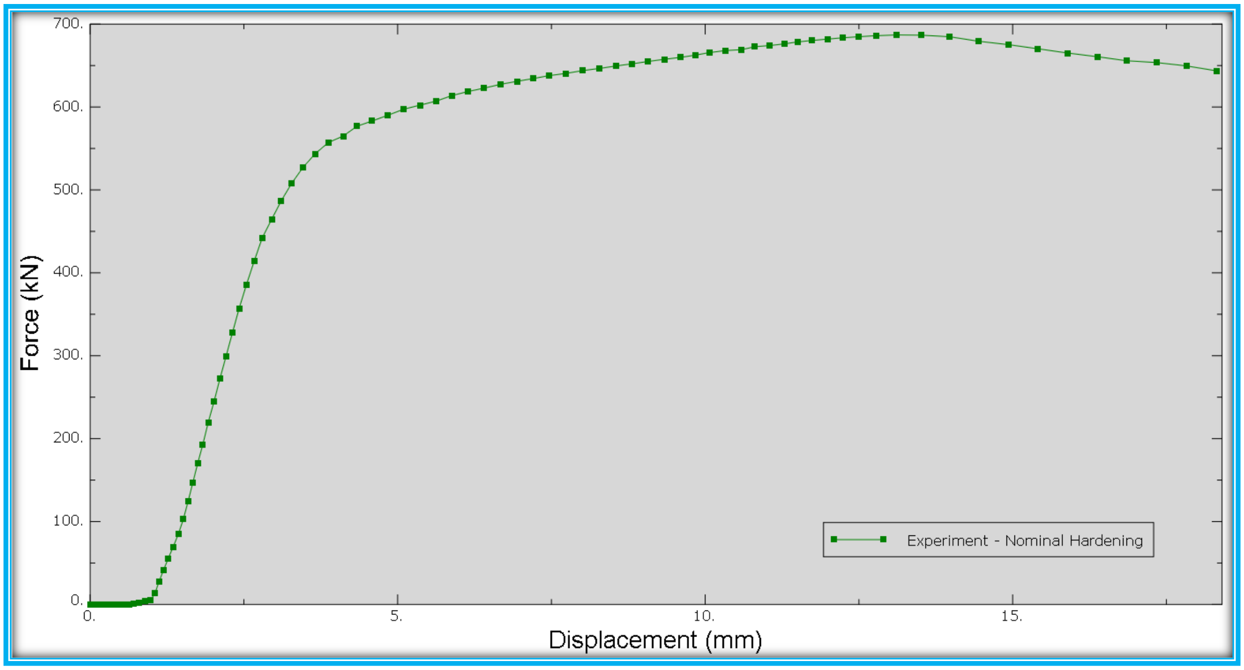


Figure 3 - 29: Experiment Joint - Force vs Displacement Curve (Nominal Hardening)

3.7.5) Comparison of Results

In order to compare the expected resistance of the experiment joint, a comparison of results from the different analysis is going to be performed. The hand calculation will be compared to IDEA StatiCA results and the ABAQUS Model using Ideal Plastic Material Model.

Analysis	Model	Shear Resistance [kN]	Failure Mechanism
Handmade Calculation (EN 1993 1-8 2020)	With Partial Safety Factos	365.50	Weld Failure
	Without Parcial Safety Factors	512.50	Weld Failure
IDEA StatiCa	5% Plastic Strain Mechanism	521.00	Weld Failure
ABAQUS	Ideal Plastic Material	551.00	Weld Failure
	Nominal Hardening Material	686.90	Weld Failure

Table 5- 8: Experiment Joint – Analyses Results

IDEA StatiCa predicted a weld resistance 1.6% higher than the new normative of the Eurocode. From this, it can be concluded that the plastic weld model of the software is able to consider the increase of the weld resistance proposed in the new code. ABAQUS Ideal Plastic Model gave 7.5% higher value for the joint resistance compared to the hand calculation.

For all the analyses, the failure mechanism is the same, failure of the weld. With this basis, the results expected from the experiment are going to give the weld behavior, which is the objective of the experiment. In addition, IDEA StatiCa gave accurate results compared to ABAQUS and the Hand calculations. This is one step forward in the validation of the software.

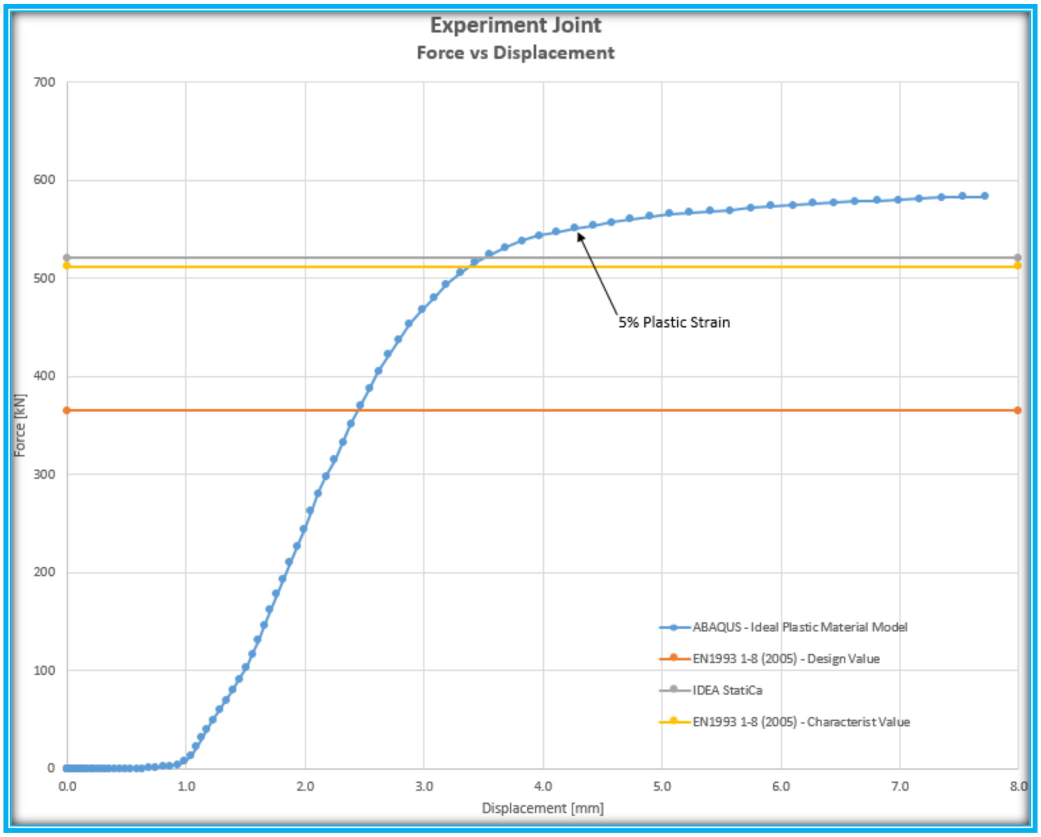


Figure 3 - 30: Experiment Joint Predictions

3.8 Experiment Results

In this thesis, only the first specimen is going to be addressed. With the remarks and information obtained from it, the experiment program can continue in proper way. Specimen 1E1 was loaded 4 times before the failure was reached. In this way, the instrumentation, frame setup and all parameters presented in the experiment were tested. Nonetheless, the results gave important remarks and prediction of the performance of the experiment and the expected results.

The three main results, which are needed for reach the goal of the experiment, will be discussed. A comparison of results, explanation of the load applied and important recommendations will be made.

3.8.1) Joint Resistance

To understand the global behavior of the joint, displacement measure dispositive (lasers) were positioned in such a way that the relative displacement between the column (HSS 200x8) and the Beam (IPE400) could be captured. This displacement is important because it is the actual displacement that the weld and the joint itself were subjected to. In addition, to validate the numerical model, the same displacement is considered.

The results in terms of Force vs Displacement curve of the final loading cycle is presented in Figure 3 - 31: Experiment - 4-load Cycles simulation. To be able to compared these results with the numerical model, the 4 load cycles were simulated using the same parameters as the Nominal Hardening analysis presented in 3.8.4. The maximum load applied at each cycle is the same maximum load applied during the experiment.

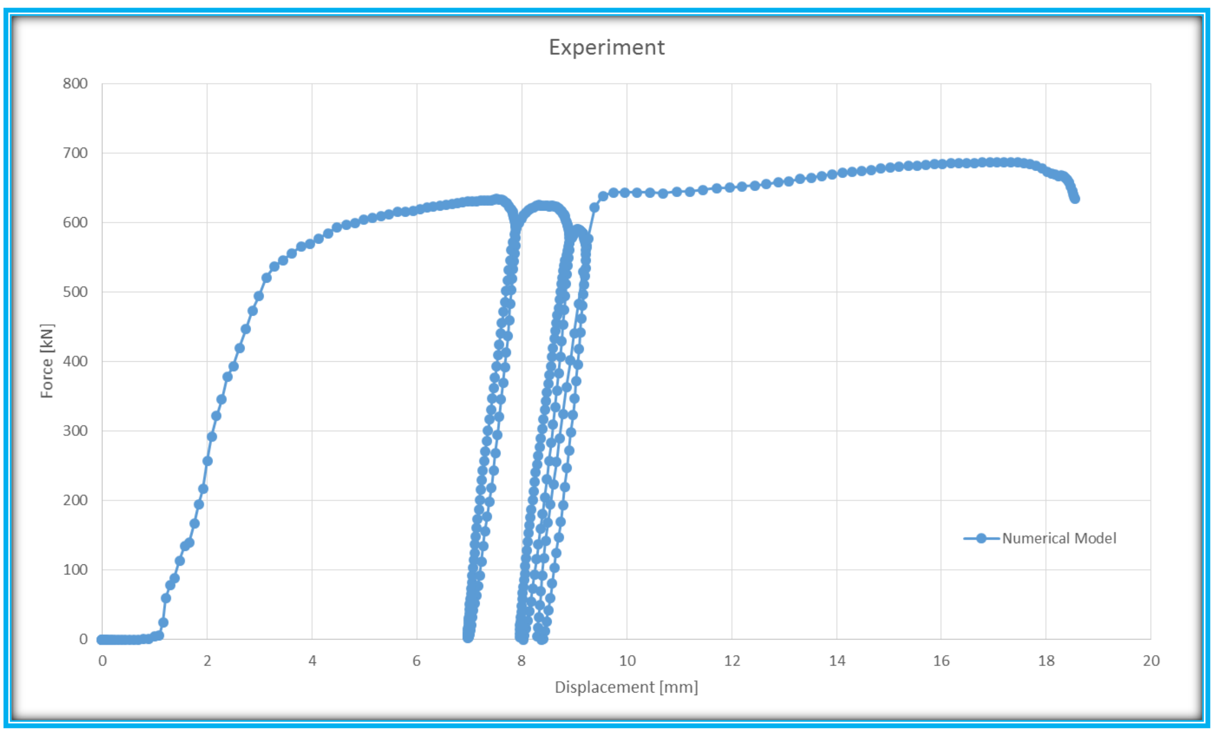


Figure 3 - 31: Experiment - 4-load Cycles simulation

It was important to simulate the 4 cycles of loading because the load applied at each cycle was bigger than the elastic maximum load for the specimen. After the first cycle, the elastic limit was increase due to yield flow caused by hardening in the joint when the plastic stage was reached. For coming cycle loads, especially the last one (which is the only one with higher load than the first cycle), the elastic behavior will happen until the elastic limit from the first load cycle is reached. This is the reason of the sharp curve found in the experiment and numerical model for the las cycle load.

The experiment results and the last load cycle from the numerical model are presented in Figure 3 – 29. It can be seen the slip in the experimental behavior. The initial stiffness corresponds to the preload of bolts (small preload of 30N*m) that after 50kN is lost and the slip of bolts happens. The bolts were located (as close is possible) in the center of the hole. For this reason, the slip behavior was possible to be captured in the experiment. After the initial stage, where rearranged of bolts and slip happened due to the hole clearance, the experimental and numerical behavior are matching. The numerical curve does not show this initial stage because it was obtained from the fourth load cycle. However, it can be observed in figure 3-28 the initial slip, which has the same magnitude of the experiment, happening in the numerical model.

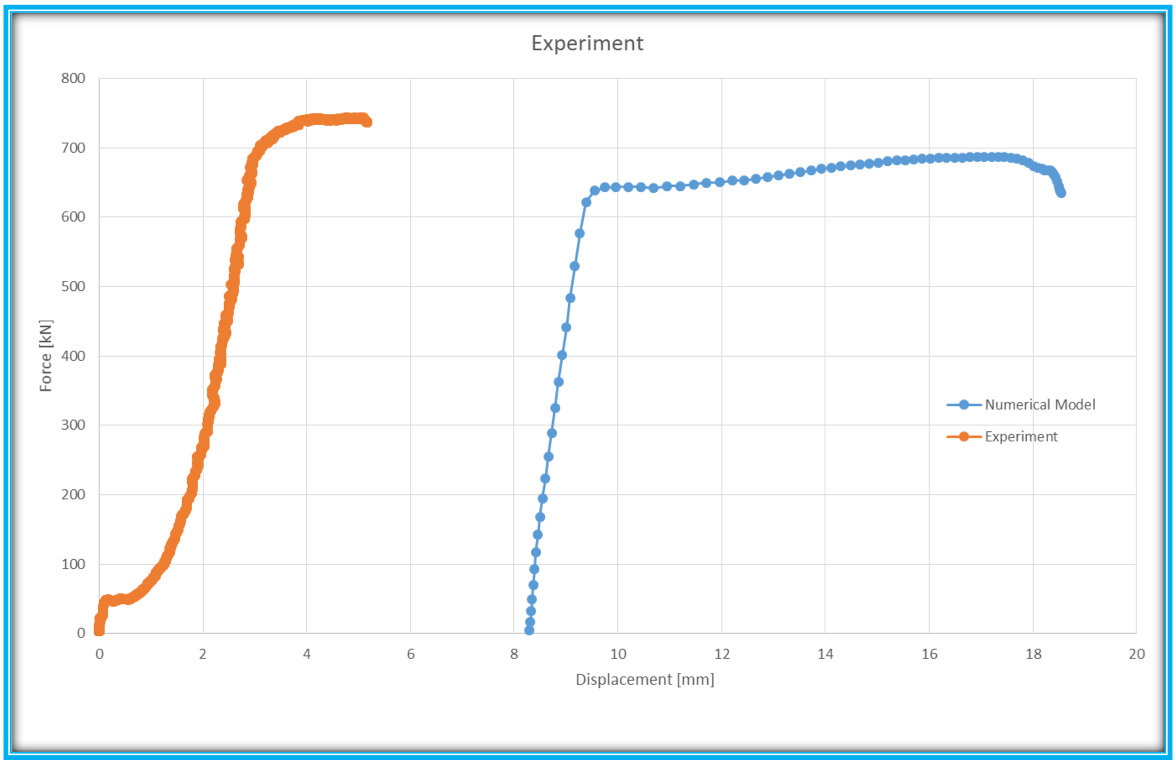


Figure 3 - 32: Numerical and Experiment Behavior of the Last Load Cycle

To have a better comparison, the displacement of the numerical model were shifted to coincide with the experimental behavior. The experimental curve started from cero because before each load cycle, the measurements were reset. Figure 3-30 shows the overlapping of the curves.

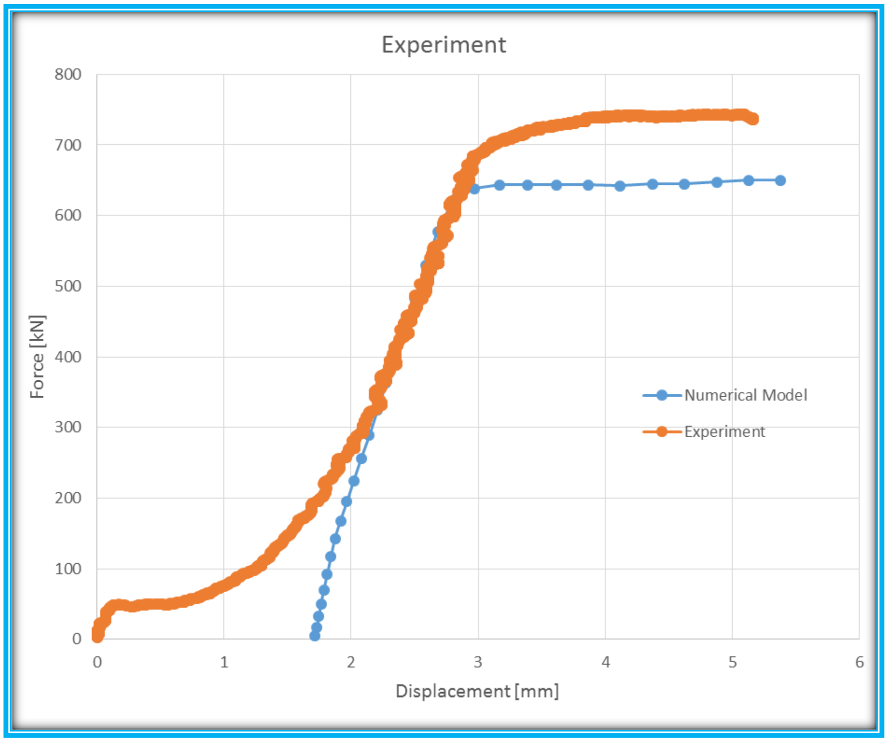


Figure 3 - 33: Experiment - Overlap of the curves

It can bee seen an accurate correspondence between the numerical model and the real behavior of the joint. The ultimate resistance reached in the experiment was of 743.5 kN. The numerical model predicted 686.5 kN, which gives a difference of 8.3%. It can be explained due to the material properties used. For the numerical, model the nominal material properties here used. However, the resistance should be compared at the same range of displacement. For this case, if we take 4mm displacement as the comparison base, a difference of 15% was found for the curves. For future correlation of the numerical model, a true stress – true strain material model should be applied. With this, the gap between both joint behavior analyses will be reduced. This will validate the numerical model in order to allow its use for future parametric uses.

Finally, figure 3-31 gives a perspective of the level of reliability present in the code, by showing the result of the different predictions against the real behavior and the numerical model. It can be seen that there is almost a factor of 2 between the design value and the experimental result.

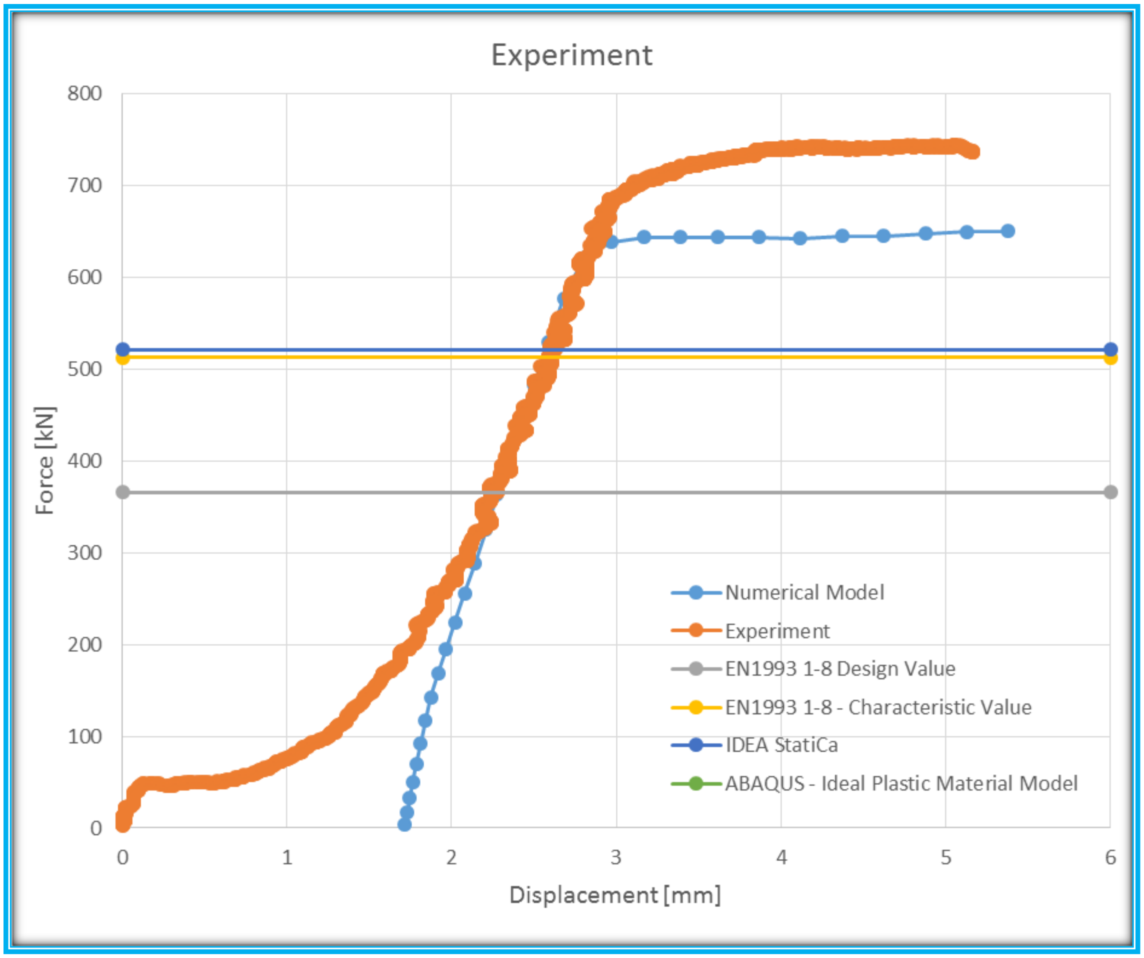


Figure 3 - 34: Comparison of Results

3.8.2) Joint Behavior

As mentioned in 3.2), one of the objectives of the experiment is to increase the knowledge about simple joints connected to flexible surfaces like the face of a hollow section. To validate the classification of simple joint, the development and distribution of bending moments in the joint will be addressed.

To achieve this goal, the real rotation center of the joint (in the elastic and plastic behavior) is going to be calculated using the numerical model from ABAQUS. In the code, the assumption of the rotational center is at the center of gravity of the bolt group. However, if the real center of rotation is further from this point, underestimating of the bending moments happening in the weld will occur. On the other hand, if the rotation center is closer to the weld the bending stress in the weld are overestimated and the resistance of the weld is underestimated.

The procedure used to find the rotation center is describe below:

1.- To choose an applied load that acts during the elastic behavior of the joint. For the plastic rotation center the ultimate load can be chosen.
2.- To measure the components of the forces happening in each bolt.

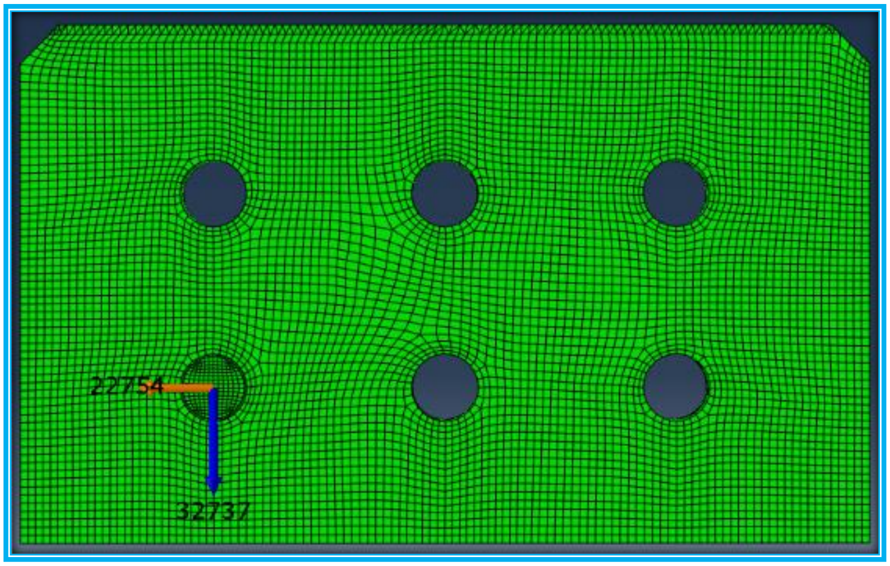


Figure 3 - 35: Components of the Second Bolt Force

3.- To draw the resultant of each bolt force using the components of them.
4.- To choose a rotation center (can be the same as the code).
5.- To measure the perpendicular distance between the resultants and the estimated rotation center.

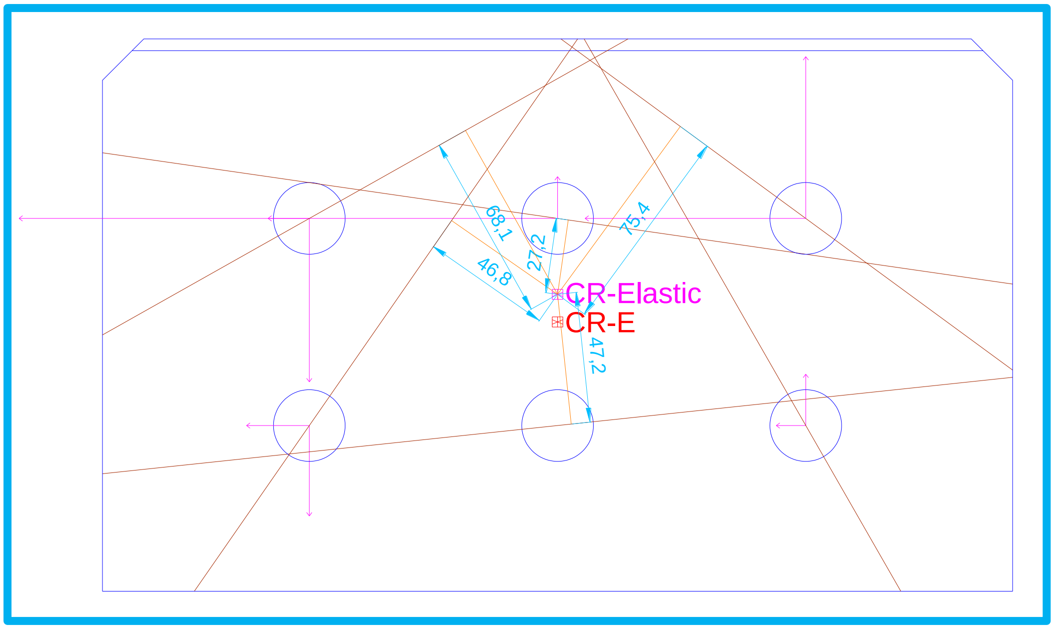


Figure 3 - 36: Perpendicular Distance between the Resultant and the Center of Rotation

6.- To calculate the moment generated by each bolt and sum them up.
7.- Just after the weld, get the bending moment in the fin plate and the shear load at that point.

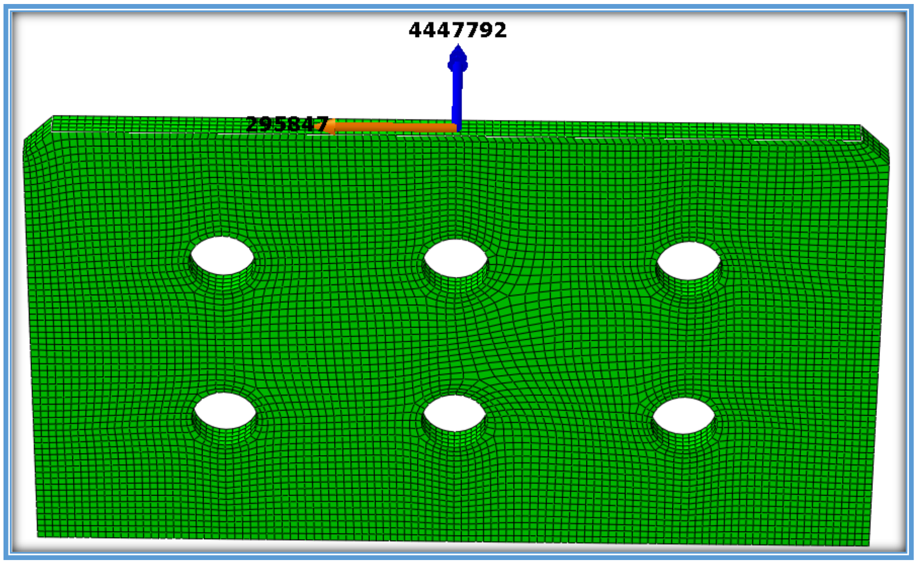


Figure 3 - 37: Shear and Bending Moment at the Fin Plate

8.- To add the Fin Plate bending moment to the moment in the bolt group

Bolt	Force X (N)	Force Y (N)	Resultant (N)	Lever Arm (mm)	Moment Y (kN*m)	Moment Resultant (kN*m)
1	105236	-59185	120737	68.1	8.22	**21.66**
2	22754	-32737	39868	46.8	1.87	
3	104938	15126	106023	27.2	2.88	
4	879	-93	884	47.2	0.04	
5	79993	58571	99144	75.4	7.48	
6	10678	18603	21450	54.4	1.17	
Fin Plate						**4.45**
Global	**295.8**		**88.25**			**26.10**
						100.0%

Table 3- 1: Elastic Rotation Center Calculation

9.- To compare this result to the moment generated from the shear load in the fin plate at the weld cross-section, using the lever arm as the distance between the assume rotation center and the cross-section. If these two moments are not the same, choose another rotation center point and repeat the steps until you get perfect match.

The same procedure was used to obtain the rotation center at the plastic stage.

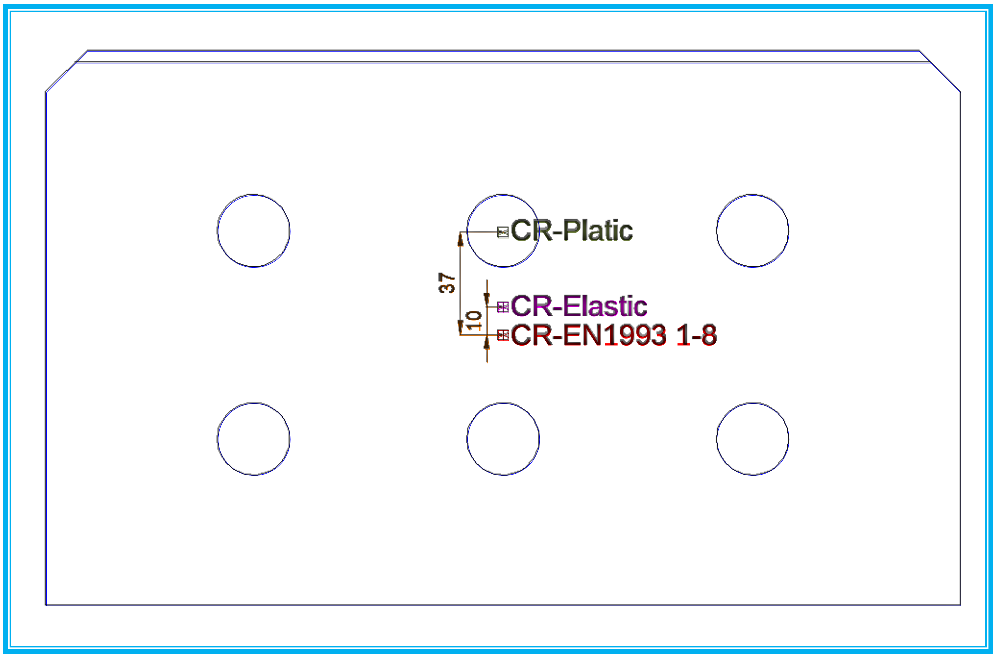


Figure 3 - 38: Real Rotations Centers

Figure 3 - 38: Real Rotations Centers shows the calculated rotation centers. It can be seen that the elastic rotation center is 10mm closer to the column face than the proposed rotation center in the code. This shows that the proposed rotation center is accurate for the elastic stage and that the location will give safe results.

In the case of the plastic rotation center, the new position is 37mmfrom the proposed rotation center. For the plastic stage, the bending moments at the welds are overestimated and the weld resistance is underestimated. This also proves that the failure in welds will happen mostly due to shear, and this explains the big gap between the code calculation and the experiment behavior. If the bending moment at the weld is not considered, the weld should fail at 702kN, which is a value closer to the actual behavior of the joint.

Another consequence of the shift of location of the rotation center in the plastic stage is the not equal distribution of forces between all the bolts. It can be seen that the first column of bolts are taking most of the load. This behavior was reflected in the numerical model and in the experiments.

3.8.3) Failure Mode and Deformation

In the same way as the prediction of the joint strength and its behavior, the deformation and failure mechanism was possible to predict with the numerical model. The failure happened due to break on the weld (see figure 3-39). The numerical model shows high plastic strains in the weld at the ultimate load as it can be seen in figure 3-40. They reached 300%, which is unrealistic but due to the material model, the strains are going to keep increasing during the yielding stage at the weld. Nevertheless, the location and concentration of the plastic strain matches with the failure mechanism observed in the experiment.

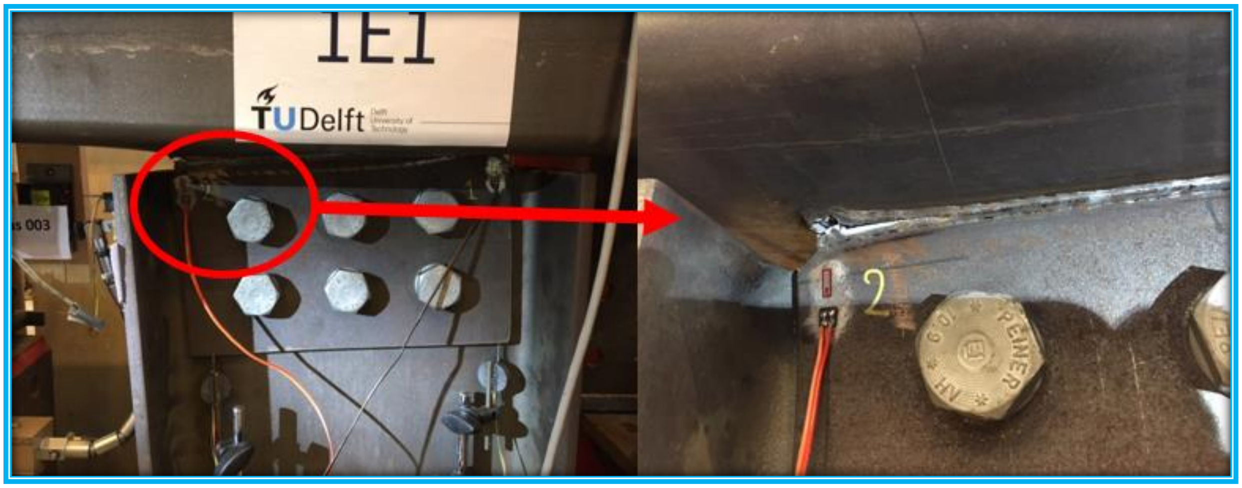


Figure 3 - 39: Failure Mechanism

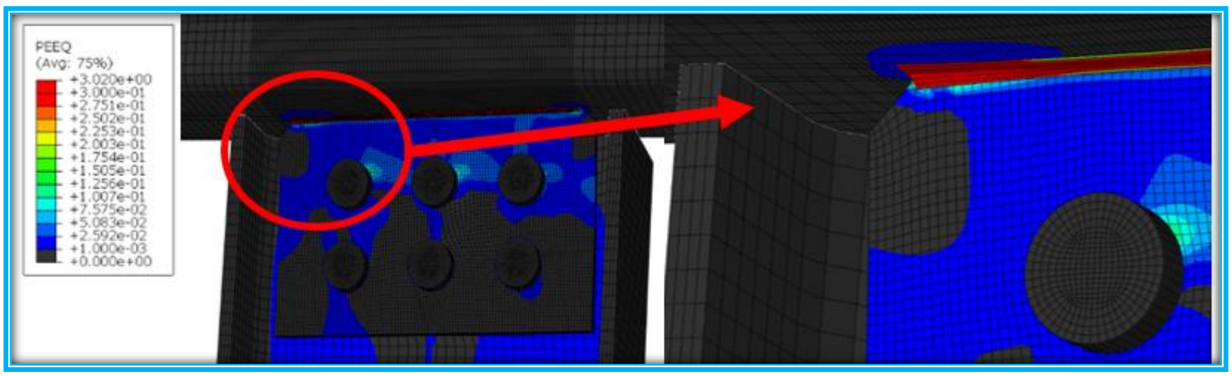


Figure 3 - 40: Plastic Strain at Ultimate Load

The bearing deformation of the holes, in the beam web and the fin plate, were accurate predicted by the numerical model (see figure 3-41). Because the deformation pattern is similar for the experiment and the numerical model, the rotation center shift due to the actual direction of the bolt forces can be validated.

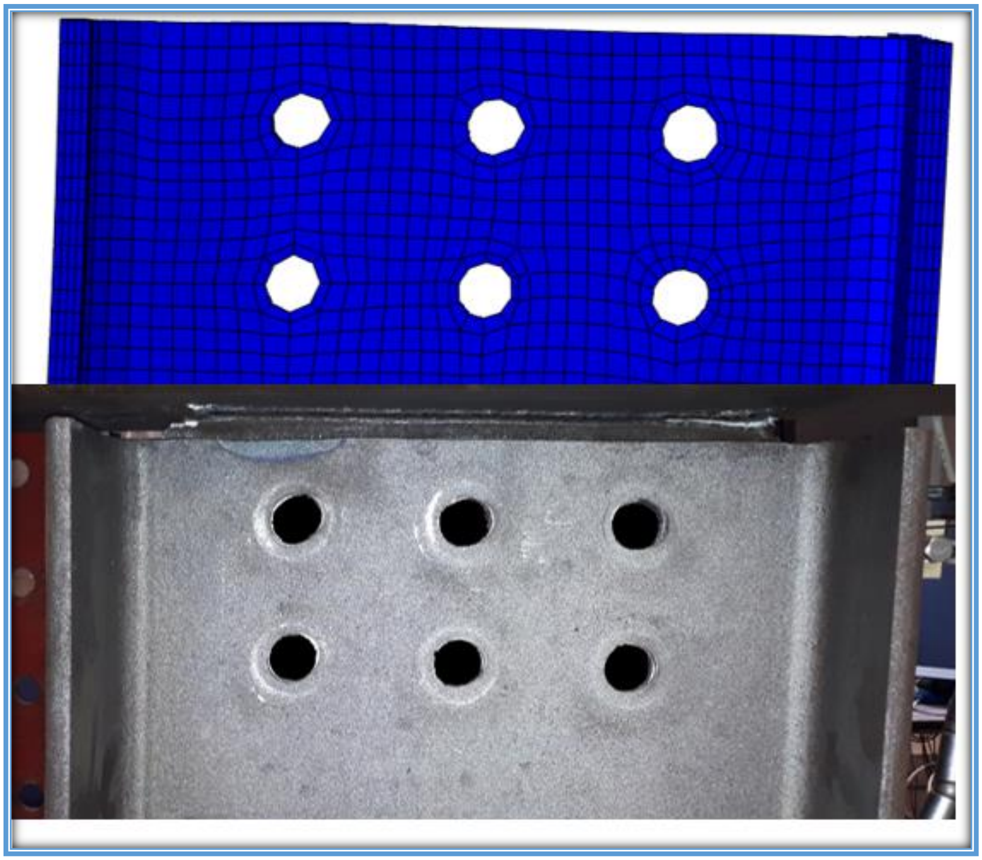


Figure 3 - 41: Hole Deformation Pattern

Chapter 4 – Hybrid Welds

4.1 Introduction

The new steel production techniques and the improvement of material properties are creating an important impact in the design of structures. Nowadays, structural designers are using the increase of the capacity of the materials to design more slender elements and bigger empty (without columns) spaces. Bridge design is also using the High Strength Steel (HSS) properties to overcome longer distances and to reduce the size of the elements. This new way to conceptualize the structures and its elements has an enormous influence when designing joints. The forces that need to be carried by the joints are increasing. Therefore, the use of high strength elements, bolts and welds, is going to be more often required.

The use of High Strength Steel can help save the amount of material used in the fabrication of a given structure, reduce the structure´s weight and the transportation costs. HSS can be efficiently used under tensile solicitations; however, stability design depends more on the elastic modulus and the inertia of the elements. For stability design the idea of using different qualities of steel for different elements, is gaining importance. Hybrid constructions, which combine HSS columns, mild steel beams (the flanges can be made of HSS) and joints with HSS components; can lead to an efficient use of steel material properties to take full advantage of its benefits.

In Chapter 4, the use of hybrid welds will be studied. With the present norms (Eurocode rules), the advantage of using high strength elements is not fully used, since the current code does not take into consideration the filler material when the load bearing capacity of the welds is calculated. Furthermore, the code suggests taking into consideration only the properties of the weaker base plate in a weld connection. These two principles are limiting the use of HSS elements with welds in under match conditions or in combination with mild steel elements. However, the new version of EN1993 1-8, which will rule the design of steel joints from 2020, already incorporate within its regulation the influence of the filler material for the weld strength design. These new rules and formulations will be analyzed below.

4.2 Difference in The Weld Strength Design EN1993 1-8 (2005 vs 2020)

In the present set of rules, the filler material properties are not considered. The weld resistance depends only on the base plate materials because only match and overmatch welds are allowed. If the weld is joining two different steel qualities, the norm stipulates that the properties from the weaker material should be used. This approach is conservative and understandable for steel grades lower than 420; however, the use of High Strength Steels is changing this way of thinking. Before, having an under match weld condition was hard to achieve because the commercial filler materials normally have higher properties than steel grades lower than S420. Nowadays, if high strength elements are used in combination with normal quality steel elements, the possibility to apply match, overmatch and under match welds is more common.

Under match welds can be advantageous in several situations. First, an under match condition can be beneficial if the weld is not critical in a high strength joint. This due to the fact that this situation allows the designer to use less resistance for the filler material. Lower resistance wires are easier to use and weld. Another situation is when a butt weld is applied between high strength plates. The tri-axial effect in the weld will allow the connection to achieve the same strength of the base materials due to the confinement of the weld (Törnblom, 2007). A third situation, the one that was applied in the experiment from chapter 3, is the use of two different base materials where one of them has mild steel properties. The resistance of the connection is influenced by the quality of the two base elements, so the use of overmatch welds not necessary means a huge increment in the resistance. The use of under match condition can be more efficient in this situation.

In terms of weld strength design, the difference between the current version of the code and the new one is reflected in the consideration of the filler material properties. The new code has two situations for the design of fillet / partial butt welds, which are listed below:

1. *Without taking into account the filler material:* In this approach, match or overmatch conditions are mandatory. It is normally applied to steel qualities S420 or lower. Here the filler material is considered to have equal or higher resistance compared to the weakest of the connection plates. This approach is the same as the approach from the present norm (2005). The difference is that the 2020 code allows the use of high strength elements and welds. In addition, the correlation factor was updated in order to incorporate HSS in the weld connection. The new values are presented below.

Table 4.1 — Correlation factor β_w for fillet welds

Standard and steel grade				Correlation factor β_w
EN 10025	EN 10210-1	EN 10219-1	EN 10149-2	
S 235 S 235 W	S 235 H	S 235 H		0,80
S 275 S 275 N/NL S 275 M/ML	S 275 H S 275 NH/NLH	S 275 H S 275 NH/NLH S 275 MH/MLH		0,85
S 355 S 355 N/NL S 355 M/ML S 355 W	S 355 H S 355 NH/NLH	S 355 H S 355 NH/NLH S 355 MH/MLH		0,90
S 420 N/NL S 420 M/ML	S 420 NH/NLH	S 420 NH/NLH S 420 MH/MLH		0,88
S 450				1,05
S 460 N/NL S 460 M/ML S 460 Q/QL/QL1	S 460 NH/NLH	S 460 NH/NLH S 460 MH/MLH		0,85
S 500 Q/QL/QL1			S 500 MC	0,90
S 550 Q/QL/QL1			S 550 MC	0,95
S 620 Q/QL/QL1			S 600 MC	1,05
S 690 Q/QL/QL1			S 650 MC S 700 MC	1,10

Figure 4 - 1: Update values for the correlation factor

The formulation of this approach is the same that can be found for the design of fillet welds in the existing code. Figure 4-2 presents the equation of the combination of stresses presented in the weld when only the base materials are taken into consideration. This formulation is only valid for match and overmatch conditions.

$$\sqrt{\sigma_\perp^2 + 3(\tau_\perp^2 + \tau_\parallel^2)} \leq \frac{f_u}{\beta_w \gamma_{M2}} \quad \text{and} \quad \sigma_\perp \leq \frac{0{,}9 f_u}{\gamma_{M2}} \qquad (4.1)$$

where

f_u is the nominal ultimate tensile strength of the part joined, which is of lower strength grade;

β_w is the appropriate correlation factor taken from Table 4.1.

Figure 4 - 2: Formulation for the first approach.

2. *Taking into account the filler material:* the second approach is the new method for the design of fillet welds considered in EN1993 1-8. It is based in several experiments done around Europe and the conclusion from a investigation program in Germany (H.P. Günther, J. Hildebrand, C. Rasche, C. Versch, I. Wudtke, U. Kuhlmann, M. Vormwald, 2009). This program concluded in some suggestions, that were accepted and used by the new version of the code. The filler material properties are taken into consideration in this new approach. Moreover, the correlation factor (β_w) was obtained from a statistical analysis of the experiments, and values linked two the properties of the filler material are given. The new values for ($\beta_{w,mod}$) are presented below:

Table 4.2 — Ultimate strength of filler metals $f_{u,FM}$ and modified correlation factor $\beta_{w,mod}$

Filler metal strength class	42	46	69	89
Ultimate strength $f_{u,FM}$ [N/mm²]	500	530	770	940
Correlation factor $\beta_{w,mod}$ [-]	0,89	0,85	1,09	1,19
For filler metals different to those given in Table 4.2 the correlation factor should be taken conservatively according to the given values.				

Figure 4 - 3: Correlation Factors for the second approach

The filler material properties are considered by modifying the mode of calculation of the ultimate stress of the weld. In this approach the ultimate stress is the combination of ¼ from the ultimate stress of the weaker base plate and ¾ from the filler ultimate strength. Figure 4-4 presents the equation for the evaluation of the weld strength under this condition. This formulation is valid for match, over match and under match conditions.

$$\sqrt{\sigma_{\perp}^2 + 3(\tau_{\perp}^2 + \tau_{\parallel}^2)} \leq \frac{0{,}25 f_{u,PM} + 0{,}75 f_{u,FM}}{\beta_{w,mod}\gamma_{M2}} \qquad (4.2)$$

where:

$f_{u,PM}$ is the nominal ultimate tensile strength of the parent metal (weaker part joined);

$f_{u,FM}$ is the nominal ultimate tensile strength of the filler metal according to , and according to EN ISO 2560, EN ISO 16834 and EN 18276;

$\beta_{w,mod}$ is a modified correlation factor that depends on the filler metal strength, see Table 4.2.

Figure 4 - 4: Formulation for the second approach

4.3 Directional Method

In the directional method the Von Mises yielding criterion is applied to an idealized triangular weld. *"…the forces transmitted by unit length of weld are resolved into components parallel and transverse to the longitudinal axis of the weld and normal and transverse to the plane of it throat"*(Eurocode3 1-8 Comitte, 2005). In other words, the stress generated by the applied force is decomposed into three stresses ($\sigma_{\perp}$, $\tau_{\perp}$ & $\tau_{\parallel}$), which are combined under the Von Mises Yielding Criterion. The three stresses and the idealized triangular shape of the welds can be seen in Figure 4 - 5: Stress Location at a Fillet Weld

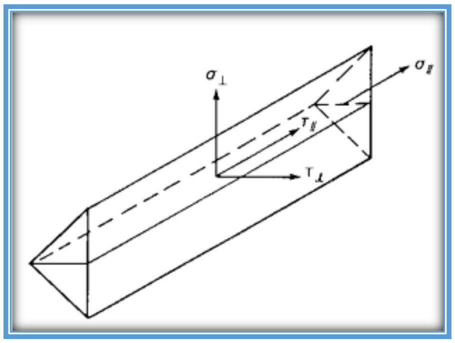


Figure 4 - 5: Stress Location at a Fillet Weld

For the idealization of the triangular shaped weld, the throat can be considered to start at the root of the weld. From Figure 4 - 6: Weld Throat thickness (a), the location of the measure of the throat thickness given by EN 1993 1-8, can be observed. Based on these criterions the directional method allow the analysis of the weld strength depending on the direction of the applied load. Tension strength is higher than shear strength. Therefore, the method allows an efficient design of the weld. However, an important assumption made is that the force is able to uniformly stress all the length of the weld (with a maximum of 150 times the throat thickness). Is typical to overserved high stresses concentrations at the ends of the weld. If the stress concentrations are too large, the performance of an elastic distribution of the stresses is recommended.

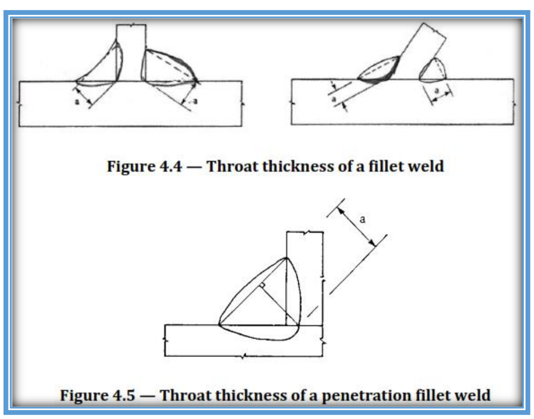


Figure 4 - 6: Weld Throat thickness (a)

Another important assumption made in the strength analysis of welds, is that if the weld is connected to two different material strength steels, the designer should use the properties from the weaker plate. This assumption is the reason why the use of high strength elements and welds in combination with normal strength plates, do not allow the design to take full advantage of the high strength properties. Futhermore, the current code is valid for steel grades until S420.

The new version of EN1993-8, takes into consideration the two points expressed before. On one side, it allows the use of high strength steels by giving an interaction factor for this kind of steels. On the other side, the code uses the suggestion given by (H.P. Günther, J. Hildebrand, C. Rasche, C. Versch, I. Wudtke, U. Kuhlmann, M. Vormwald, 2009) (see chapter 2), which takes into consideration the combination of the filler material and the base plates materials. The code gives the interaction factor for four different filler material strengths. Nevertheless, the suggestion of using only the properties from the weaker base material, is still present.

Below will be presented the difference of the weld connection capacity for shear and tension load per unit length. Additionally, a practical example will be performed to see the difference in the strength calculations for a Hybrid Weld. Equations 4.1) and 4.2) will be used.

Tension Resistance: $$F_{T,RD} = \frac{\sqrt{2} * a * f_{u,w}}{2 * \beta_w * \gamma_{M2}} \tag{4.3}$$

Shear Resistance: $$F_{S,RD} = \frac{a * f_{u,w}}{\sqrt{3} * \beta_w * \gamma_{M2}} \tag{4.4}$$

where, $F_{T,RD}$ = Tension Resistance Force of the weld per unit length

$F_{S,RD}$ = Shear Resistance Force of the weld per unit length

a = Throat Thickness according to Figure 4 - 6: Weld Throat thickness (a)

β_w = Interaction Factor given by figure 4-1 or 4-2

γ_{M2} = Partial Safety Factor (1.25)

$f_{u,w}$ = Ultimate strength of the weld (see 4.2)

The weld strength will be calculated for a tensile and shear loading. "T" specimens of 50mm with 10mm and 20mm thickness will be used in a weld with 3mm thickness (see Figure 4 - 7: Weld Specimen). The filler material proposed, is a commercial wire (Mega Fill 742M) for welding HSS elements. Its yielding resistance is 690 [N/mm^2] and its Ultimate strength is 800 [N/mm^2].

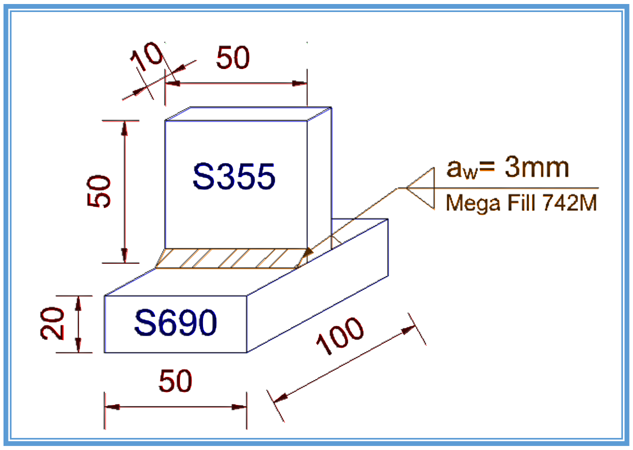


Figure 4 - 7: Weld Specimen

The difference in the weld strength resistance according to the present code and the new code is showed below:

Code	2* ℓ_{eff} [mm]	$f_{u,w}$ [N/mm²]	β_w	a [mm]	γ_{M2}	Tensile Resistance [kN]	Shear Resistance [kN]
EN 1993 1-8 (2005)	100	510.0	0.95	3.00	1.25	91.11	74.39
EN 1993 1-8 (2020)	100	727.5	1.09	3.00	1.25	113.27	92.48
EN 1993 1-8 (2020) - Variation	100	760.0	1.09	3.00	1.25	118.33	96.61

Table 4- 1: Weld Design Strength

The third option (EN1993 1-8 (2020) – Variation) is similar to the new meth present in the new version of the code but considering the strength of both base plates. Each plate has an influence of 12.5% of the final ultimate strength of the weld ($f_{u,w}$).

Table 4-1 presents the difference in the weld strength design value. An increase of the resistance can be seen. By taking into consideration the new code, the same High Strength Weld (HSW) can increase its calculated capacity in a 24.3%. Moreover, if the limitation of the code for only use the material properties of the weaker base material is avoid, the weld capacity can increase a 29.9%.

The significant increase in weld resistance can lead to important savings in time and welding costs due to the reduction of the weld size.

4.4 Simplified Method

The efficient results obtained from the directional method can be overshadowed by the detailed (time consuming) analysis required. The code gives an alternative to this method by applying the so called "Simplified Method". The advantage of this procedure is that the designer does not need to take into consideration the direction of the applied load. Equation (4.2) can be directly used and compare its result to the stress caused by the applied load per unit length. In other words, only the shear resistance, which is smaller than the tensile resistance, is taken into consideration for the analysis of the weld strength. The values of β_w and $f_{u,w}$ are the same as the ones applied in the Directional Method (see 4.2). The triangular idealization of the weld and the location of the throat thickness are also the same for both methods.

The design of weld resistance can be assumed sufficient if at any point of the length of the weld, the resistance per unit length is bigger or equal to the resultant of all the forces effects, per unit length.

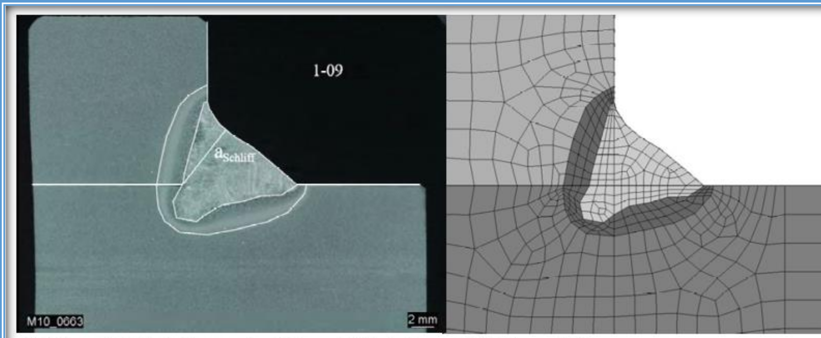


Abbildung 9.1: Modellierung der Makroschliffe in Ansys 13.0® [Ansys 13.0]

Figure 4 - 8: Complex Finite Element Model of Welds (Rasche & Kuhlmann, 2012)

4.5 Numerical Analysis

In the previous sections of the chapter the analytical formulation for welds strength analysis was presented. However, the equations proposed by the Eurocode commonly are in the safe side. Due to complex geometries and all the different zones properties of the weld, like the heat affected zone, an exact formulation is a complex analysis to perform. Using Finite Element Analysis is an option to overcome this problem. The use of FEA is a suitable approach for this task and already has been proved to give accurate predictions of the weld resistance when models are compared to experiments (Rasche & Kuhlmann, 2012).

Nevertheless, the same weld model that will be use in chapter 5, will tested in order to observe the accuracy of the model to predict the characteristic value of the weld strength.

The weld model consists in a rectangular-isosceles triangle with the height (measure perpendicular to the hypotenuse) equal to the throat thickness (a). The sides of the weld are tided to the base plates. A perfect plastic material model is used with the yielding stress equal to the ultimate weld stress (f_u). The ultimate stress is the nominal ultimate stress of the filler material. The base plates also have a perfect plastic material model with the yielding stress equal to the nominal yielding stress of the material.

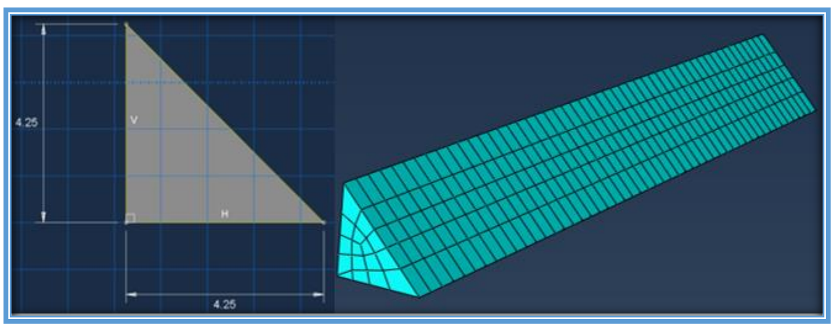


Figure 4 - 9: Weld Model – Throat thickness 3mm (ABAQUS)

4.5.1) Welds Loaded in Tension

The same practical example from part 4.3 is used to compare the weld model of the numerical analysis with the characteristic values calculated with the codes rules. This is the case of a hybrid weld, the same situation that was tested in chapter 3. The failure criterion is the 5% plastic strain mechanism that is going to be use in chapter 5 for the global failure criterion of joints. In addition, a weld size effect analysis will be performed, to check the influence of these parameter in the numerical model.

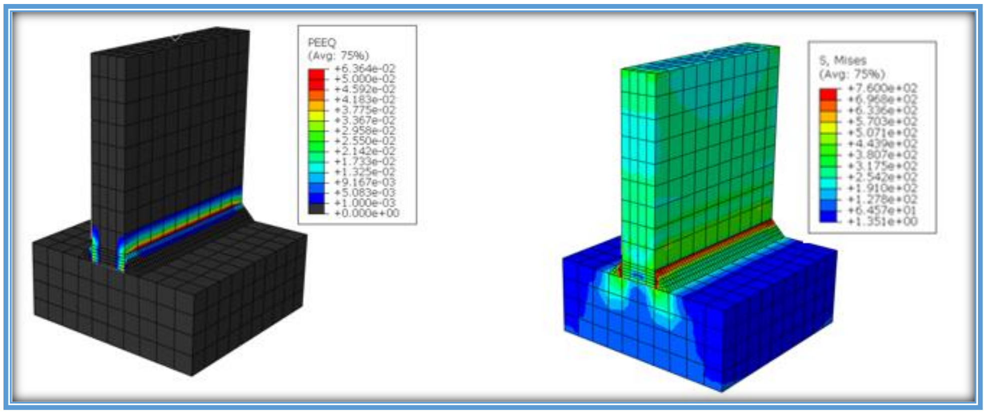


Figure 4 - 10: Strain/Stress Distribution in the Tension Analysis

Code	$2*\ell_{eff}$ [mm]	$f_{u,w}$ [N/mm²]	β_w	a [mm]	γ_{M2}	Tensile Resistance [kN]
EN 1993 1-8 (2005)	100	510.0	0.95	3.00	1.00	113.88
EN 1993 1-8 (2020)	100	727.5	1.09	3.00	1.00	141.58
Numerical Analysis - Abaqus	100	800.0	-	3.00	-	131.50

Table 4- 2: Tensile Loaded Weld Results

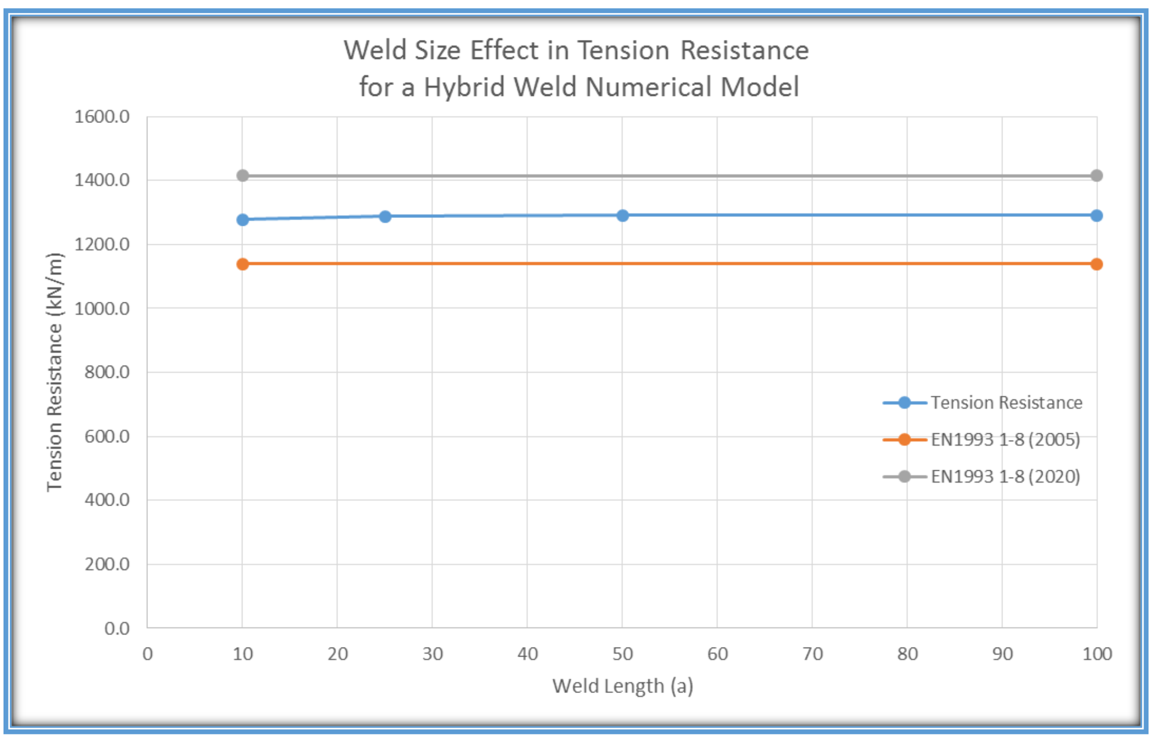


Figure 4 - 11: Results Comparison for the Tension Loaded Weld

Table 4- 2: Tensile Loaded Weld Results showed the results from the numerical analysis of a tension-loaded hybrid weld, against the characteristic values obtained from the two Eurocodes. The numerical prediction is 7.12% lower than the new code and 15.55% higher than the current code.

The weld model predicted the tensile strength of the weld with results that are in the safe side. This means that the model underestimate the strength of the weld compared to the new formulation of the Eurocode. Nevertheless, the result shows an increment from the actual code, which means that the properties of the filler material are used for a more efficient design.

The size effect analysis shows no influence in the weld strength when it is loaded in tension.

4.5.2) Welds Loaded in Shear

In the same way as the tensile-loaded weld, a shear-loaded weld will analyzed. The same criterion of 5% plastic strain mechanism will be applied.

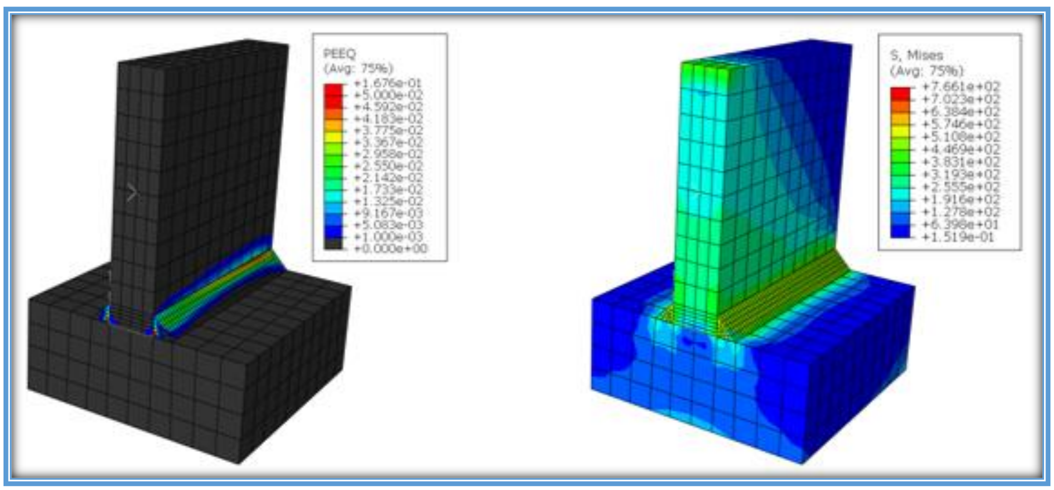


Figure 4 - 12 Stress/Strain Distribution in the Shear Loaded Weld

Code	$2^* \ell_{eff}$ [mm]	$f_{u,w}$ [N/mm²]	β_w	a [mm]	γ_{M2}	Shear Resistance [kN]
EN 1993 1-8 (2005)	100	510.0	0.95	3.00	1.00	92.98
EN 1993 1-8 (2020)	100	727.5	1.09	3.00	1.00	115.60
Numerical Analysis - Abaqus	100	800.0	-	3.00	-	89.45

Table 4- 3: Shear Loaded Weld Results

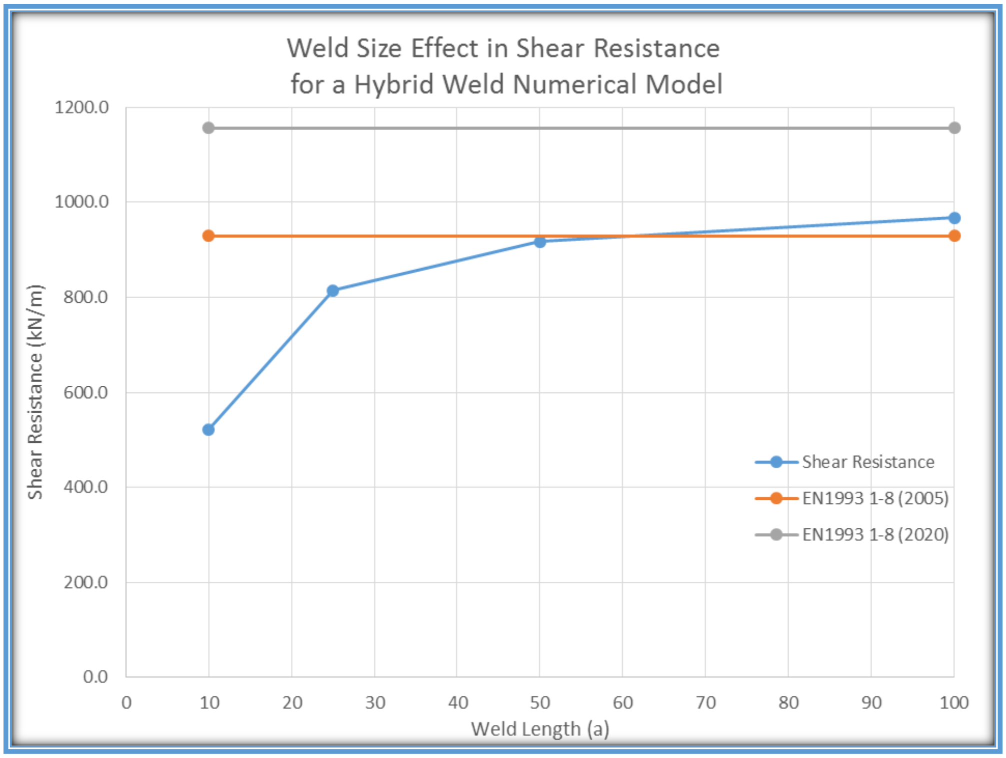


Figure 4 - 13: Results Comparison for the Shear Loaded Weld

When the weld is loaded in shear, the accuracy of the numerical model is different. In this case, the results are similar to the ones obtained from the previous code. The results have a value 22.62% lower than the calculated by the new code and less than 4% lower than the calculated by the present code. The weld model that will be use in chapter 5, gives conservative values when is subjected to pure shear.

In addition, the size effect analysis showed an important negative effect to the strength of the weld when is subjected to pure shear. Weld models with lengths smaller than 50a, will give inaccurate prediction of the weld strength. This parameter need to be taken into consideration during a numerical analysis.

If the FEA is used for a hybrid weld that is loaded in shear and tension, the results will be higher than the current code, but conservative compared to the new code. From this result, it can be concluded that the proposed weld model can be used for the finite element model of joints that will be used in chapter 5. The results are going to be conservative but the design will be more efficient than using the actual code formulation.

Chapter 5 – Joint Design Assessment

5.1 Introduction

In previous chapters, the accuracy of numerical and analytical prediction of the real behavior of a steel joint was evaluated. Great Similarities between the numerical and the analytical results were achieved. Later on, the experiment showed how accurate can these predictions be. The level of safety and reliability regulated in the Eurocode rules were also possible to witness. Nevertheless, chapter 5Th will present results obtained from different analysis applied to a set of steel joints. These results can be compared and validated. The aim of this study is to analyze how finite elements tools and the Eurocode formulations can accurately predict the capacity of steel joints, which are easily found in daily practice of structural engineering. During this comparison, the finite element analysis will be performed using two different software. The first is ABAQUS, an experimental software widely used. The second one is IDEA StatiCa, a daily practice software that will be used and validated.

The study will be performed by analyzing one joint for each type of steel joint stipulated in the Eurocode classification according to its stiffness. The first is a simple joint (pinned) whose numerical model is similar to the experiment joint presented in chapter 3. The other two are a rigid joint and a semi-rigid joint taken from SERICOM-II database. SERICOM is a database whose objective is to collect the information obtained from experiments all around Europe. The main goal is to document and to generate a catalog of the behavior of diverse kinds of semi-rigid joints. Within this database, the experimental behavior and the real measures of these joints are included.

5.2 Conditions of the comparison

Due to the fact that different types of analysis and tools are going to be used in this comparison, a base of principles and parameters need to be set in order to have a proper contrast of the results. The use of safety factors, the material model, the failure limit, among others; are parameters that need to be set to have a fair judgment. The base marks are presented below:

- *Material Model:* Ideal Plastic Material Model will be used to model the steel elements. This model is similar to the assumptions made in the mechanics formulation of the different failure modes and components. The formulation normally takes into consideration the yielding capacity of the material; however, they do not take into consideration the plastic strain on it. Furthermore, strain hardening is completely neglected by the formulation. When a component does not have the deformation capacity to redistribute the yielding stresses, the ultimate stress is used but with a higher partial safety factor. An example of this is the shear block failure of a shear resistance joint. Here, the area in tension uses the yielding stress, but the area in shear uses the ultimate stress. The reason behind it is the deformation capacity of the different components. An advantage of using this material model in the analysis is that IDEA StatiCa uses it (see A.2.3). For these reasons, this material model will also be used in ABAQUS.

  A second material model will also be used. A Nominal Hardening Model will be specified in ABAQUS in order to compare how accurately this analysis can predict the real behavior of the joint which is given by the information in SERICOM database.

- *Failure Limit:* When a numerical model is used to evaluate the resistance of a steel joint, obtaining the maximum load capacity or the point in which the failure mechanism start to happen, is not always straight forward. When a simple element is modeled and this element yields, a horizontal plateau will be found in the force vs displacement curve. However, in steel joints, different components start yielding in different time and in a different way. Local yielding occurs and a redistribution of forces and stress happen between the components. This provokes a big curve zone after the elastic zone and before reaching the yielding plateau. The "big curve" part can be seen in the "Moment vs. Rotation curve" (see figure 5-17). When this kind of behavior is present, big plastic strain can occur before the horizontal yielding plateau happens. For this reason, and using the pre-set parameter from IDEA StatiCa, a limit of 5% of plastic strain will be used to set the failure point of the joints. The 5% is also recommended by the Eurocode (EN 1993 1-5, Annex C).

- *Safety Factors:* During a numerical analysis or in an experiment procedure, safety factors are not presented. When the joint resistance is calculated by using these factors and other ones like interaction factors, the result is called the design resistance of the element/joint. Nevertheless, to compare experimental and numerical results with the mechanical formulation for the resistance of the different components and elements, the characteristic value needs to be computed. This value can be obtained by not using the safety factors.

Nonetheless, the results “with” and “without” are presented for the hand calculations. This due that the design value is the one that should be used in daily engineering practice.

- *Modeling Properties:* The boundary conditions, the contact and the geometry of the joints were modeled as similar as possible between the experiments, the semi-analytical formulation and the available modeling options in the two numerical software. The boundary conditions try to resemble the assumption made in the experiments and in the hand calculations (Eurocode rules). The most difficult elements during the joint modeling are the brittle elements like bolts and welds. The bolt model used is the same solid model used in chapter 3 (Kim, Yoon, & Kang, 2006). This model shows good results when strength is calculated. For the stiffness of the joint, a pretension load and a better detailing of the nut, head and thread part are needed if accurate stiffness prediction is required. If the failure mechanism is happening due to shear or tension failure of the bolt, it is hard to get accurate results from this bolt model. Nevertheless, the simplicity of this model is required during this study and during engineering practice. Bolt failure should be avoided.

- *Geometry:* For some of the experiments the geometry is not precise. The lack of pictures and information due to the long time elapsed since the experiment was carried out is another limitation to model the exact joint. When the information from SERICOM database was not enough or was misleading, a logical assumption of the geometry was performed. The same geometry was used for all the types of analysis (hand calculations, Idea StatiCa and ABAQUS software). The actual (measure) dimensions were used; however, where there was lack of information, the nominal dimensions and properties were used.

- *Design Resistance:* For practical designs, the design resistance is needed. This value is calculated by using the Eurocode formulations with all its factors. For the purpose of using IDEA StatiCa in daily practice, a set of parameters is proposed in order to obtain the design values. The plastic strain should be limited to 3% and the number of elements at the critical edge (see Annex A), should not be less than 24. Moreover, the finite element size should be set in the rage between 20[mm] and 10[mm]. Following these parameters and using the Stiffness Analysis, the design resistance can be easily obtained.

5.3 Simple Joint (Pinned)

5.3.1) Geometry Description

The first type of joint that will be used to compare the results of the different analysis is a simple joint. This joint was taken from the examples given in the Steel 2 Lecture within the Master Program of Structural Engineering at TU Delft. Nominal values are used for the properties of the materials and the geometry of the profiles. Even when the mentioned joint is not a direct experiment that can be used to validate the results, it is similar to the joint showed in the experiment performed for this dissertation (see Chapter 3). For this reason, the numerical models and the hand calculations were already validated at chapter 3.

A simple joint is a shear resistant joint that normally is idealized as a hinge that has no bending resistant. Chapter 3 probes that some bending stress is happening in the joint, for this reason, the eccentricity and other parameters need to be taken into consideration. For instance, the bending resistance of the column web (where the connection is happening). This simple joint is a fin plate bolted joint, which is welded to the column web by a double side fillet weld. The load is applied exactly at the bolt column. Figure 5- 1: Simple Joint - Geometry Detail shows the geometry details of the joint.

Element		Column	Beam	Fin Plate	Bolts	
Profile		HEA 200	IPE 300	PL230x100x8	M20 - 8.8	
Dimensions [mm]	*Length*	-	-	-	20.00	Bolt diameter (tension) [mm]
	Heigth	190.00	300.00	230.00	13.00	Thickness of bolt nut [mm]
	Width	200.00	150.00	100.00	36.95	Across points dimension of bolt head & nut
	Flange Thickness	10.50	10.70	8.00	16.00	Thickness of bolt head [mm]
	Web Thicknes	6.50	7.10	-	4.00	Thickness of ALL washers [mm]
	Rolling Radius	18.00	15.00	-	4.00	Fillet Welds Throat
Material Properties [kN/mm²]	*Yielding Stress (fy)*	235.00	235.00	235.00	640.00	
	Ultimate Stress (fu)	360.00	360.00	360.00	800.00	
	Young Modulus	210000.00	210000.00	210000.00	*210000.00	

Table 5- 9: Simple Joint Material and Geometry Properties

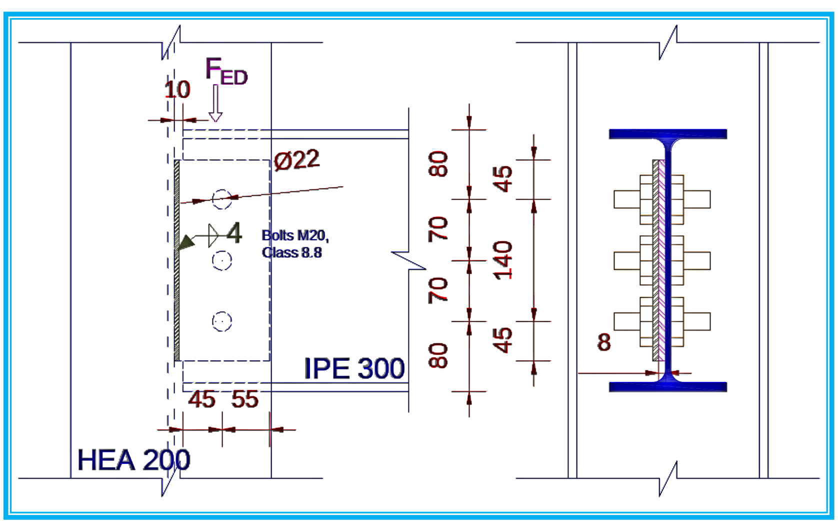


Figure 5- 1: Simple Joint - Geometry Detail

5.3.2) Hand Calculation (Eurocode Rules)

The design resistance of the joint calculated by using the Eurocode formulations presented in the guidelines (Jaspart, 2016), was of 144.5 [kN] (see Annex D). The failure mechanism is the bearing of the beam web. Nevertheless, shear resistance of the bolts and the bearing resistance of the fin plate have a little higher shear resistance (5.34% and 11.92% respectively).

If the characteristic shear resistance of the joint is calculated, the partial safety factors, interaction factors and eccentricity factors must not be used. In these calculations, a joint resistance of 219.62 [kN] is obtained. The failure mode in this analysis is also the bearing resistance of the beam web. Normally this kind of failure is hard to obtain in a real size joint experiment, and is possible that the shear of the plate is present in the failure mode. Nevertheless, even when numerical models are able to capture this failure mode, the material model plays an important role in the accuracy of the strength resistance. The summary of the calculation of the different components is shown below:

Failure Mode		**EN 1993 1-8 (2005) [kN]**	
		With Safety Factors	**Without Safety Factors**
$V_{RD.1}$	Shear Resistance of Bolts	152.17	243.91
$V_{RD.2}$	Bearing Resistance of the Fin Plate	161.64	267.31
$V_{RD.3}$	Shear Resistance Fin Plate: Gross Section	196.57	249.65
$V_{RD.4}$	Shear Resistance Fin Plate: Net Section	218.16	272.69
$V_{RD.5}$	Shear Resistance Fin Plate: Shear Block	186.03	253.42
$V_{RD.6}$	Bending Fin Plate	301.37	301.37
$V_{RD.7}$	Buckling Fin Plate (Factor)	301.37	301.37
$V_{RD.8}$	Bearing Resistance of the Beam Web	144.43	219.62
$V_{RD.9}$	Shear Resistance Beam Web: Gross Section	348.28	348.28
$V_{RD.10}$	Shear Resistance Beam Web: Net Section	348.91	436.14
$V_{RD.11}$	Shear Resistance Beam Web: Shear Block	193.71	245.85
$V_{RD.12}$	Shear Resistance Column Web	973.62	1217.03
$V_{RD.13}$	Welds Resistance	336.90	421.89

Table 5- 10: Simple Joint Failure Modes Shear Resistances

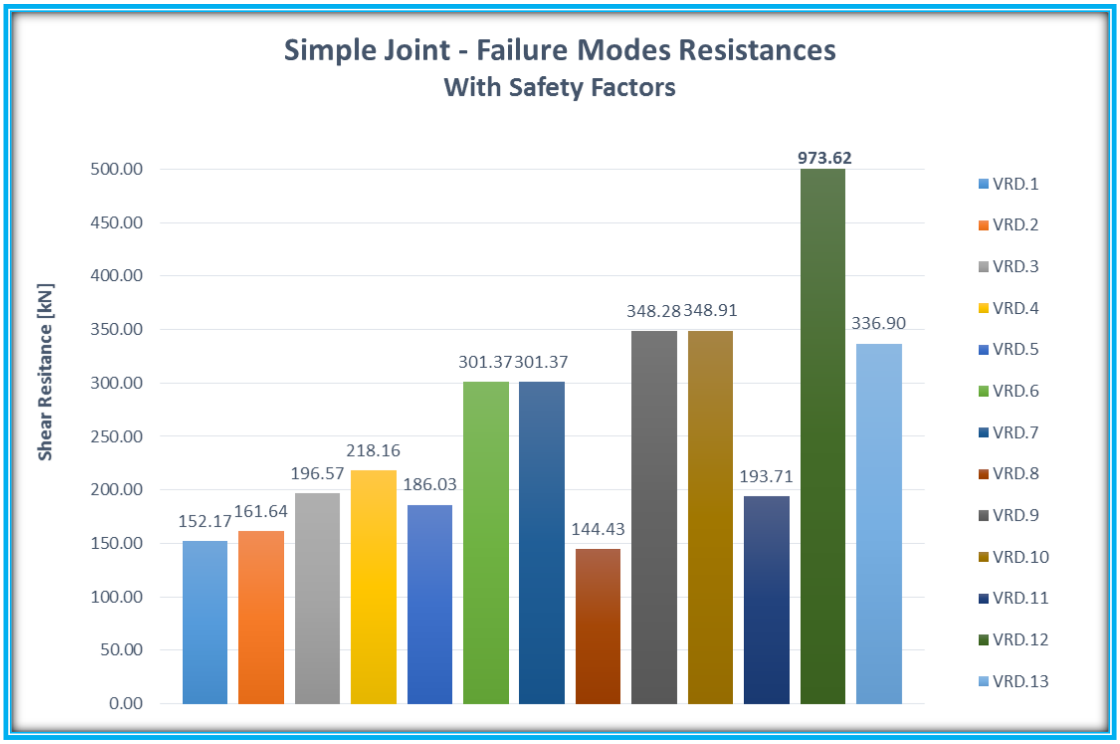


Figure 5- 2: Simple Joint - Hand Calculation Results

This simple joint is a standard joint, not only in its geometry but also in its properties. COP software was used to compare its results with the hand calculations. Figure 5- 3: Simple Joint - COP Results shows COP summary of results. The values are the same as the values from the hand calculation following the Eurocode guidelines.

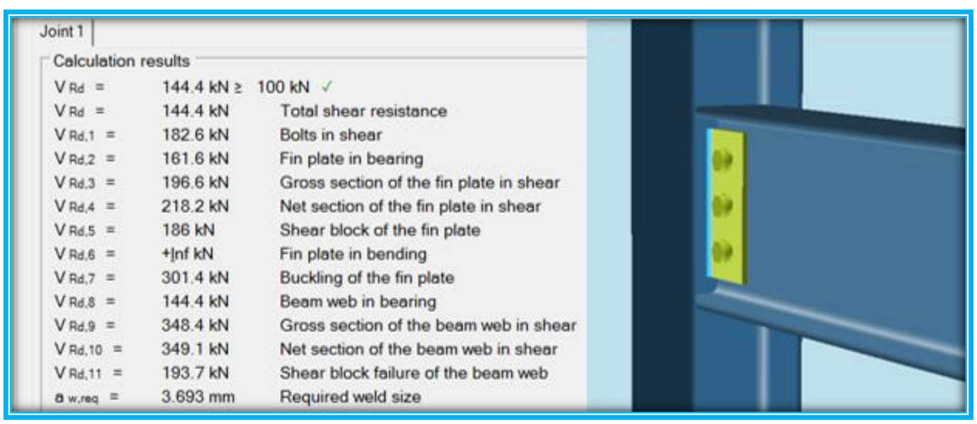


Figure 5- 3: Simple Joint - COP Results

5.3.3) IDEA StatiCa Analysis

The stiffness analysis cannot be used for a shear resistant joint. Therefore, only a Stress/Strain Analysis was performed using IDEA StatiCa software. The limit for the analysis was chose when a 5% of plastic strain mechanism was presented in any of the plates. The shear resistance calculated by the software was of 97.00 [kN]. The failure mechanism is plasticization of the column web.

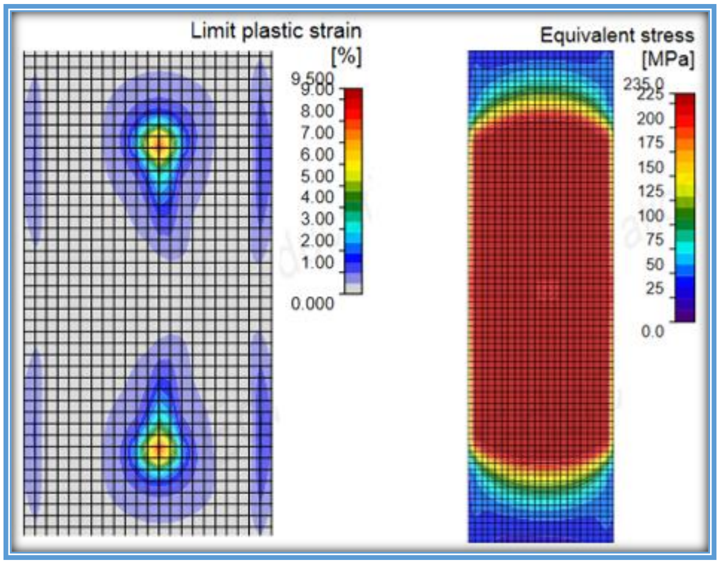


Figure 5- 4: Simple Joint - Strain/Stress distribution at the Column Web

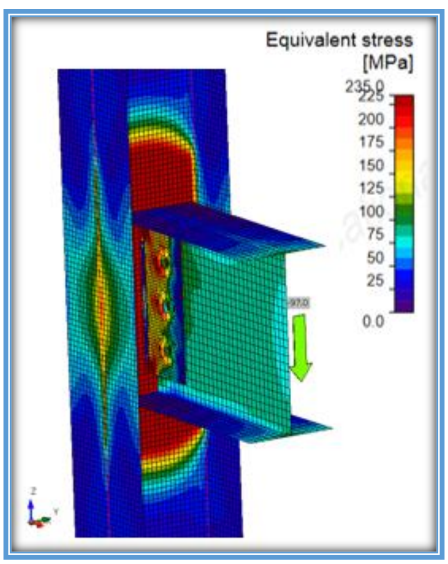


Figure 5- 5: Simple Joint - Stress Distribution (IDEA StatiCa)

5.3.4) ABAQUS Analysis

For the ABAQUS numerical analysis, by using the Ideal Plastic Material Model and by taking into consideration the 5% Plastic strain limitation, the shear resistance of the joint was calculated in 215.20 [kN]. The failure mode that is limiting the joint resistance is the Bearing of the Fin Plate. Nevertheless, Figure 5- 7: Simple Joint - Stress Distribution (ABAQUS Ideal Plastic Material Model) shows yielding at the column web as well at the shear block failure for the fin plate. This suggests that other failure modes are present in the failure mechanism; however, the critical one was chosen by looking which one of the failure modes reaches first the 5% plastic strain mechanism.

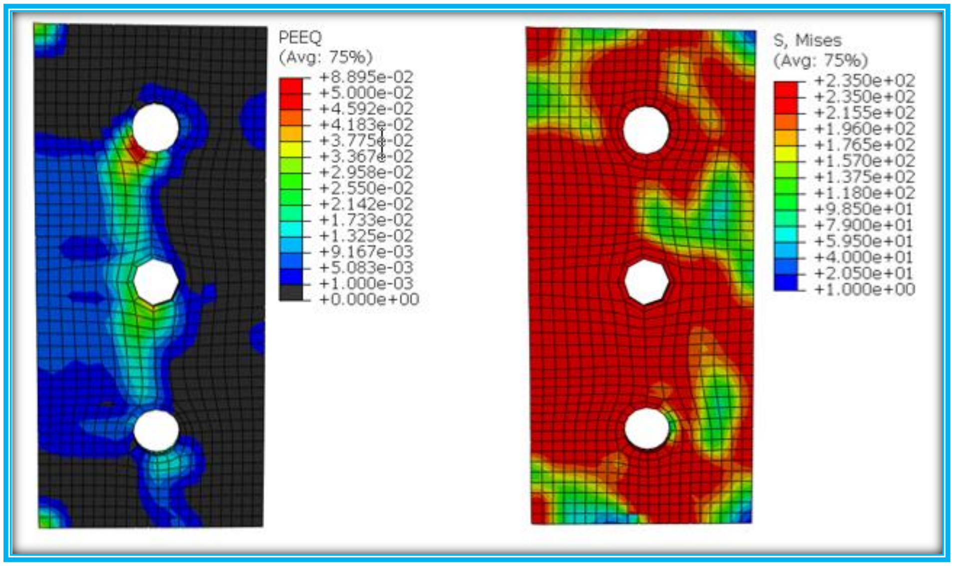


Figure 5- 6: Simple Joint - Strain/Stress Distribution of the Fin Plate (ABAQUS)

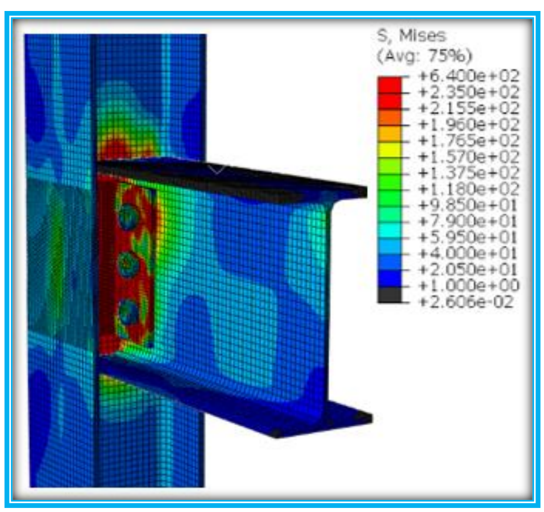


Figure 5- 7: Simple Joint - Stress Distribution (ABAQUS Ideal Plastic Material Model)

The Force vs Displacement curve presented in Figure 5- 8: Simple Joint - Force vs Displacement Curve (ABAQUS)shows the behavior of the joint. The limit condition happened in the inelastic part of the curve, which was already expected due to the type of material model used in ABAQUS. It also can be seen that at the beginning no force is developed, this occurs due to the bolt hole clearance.

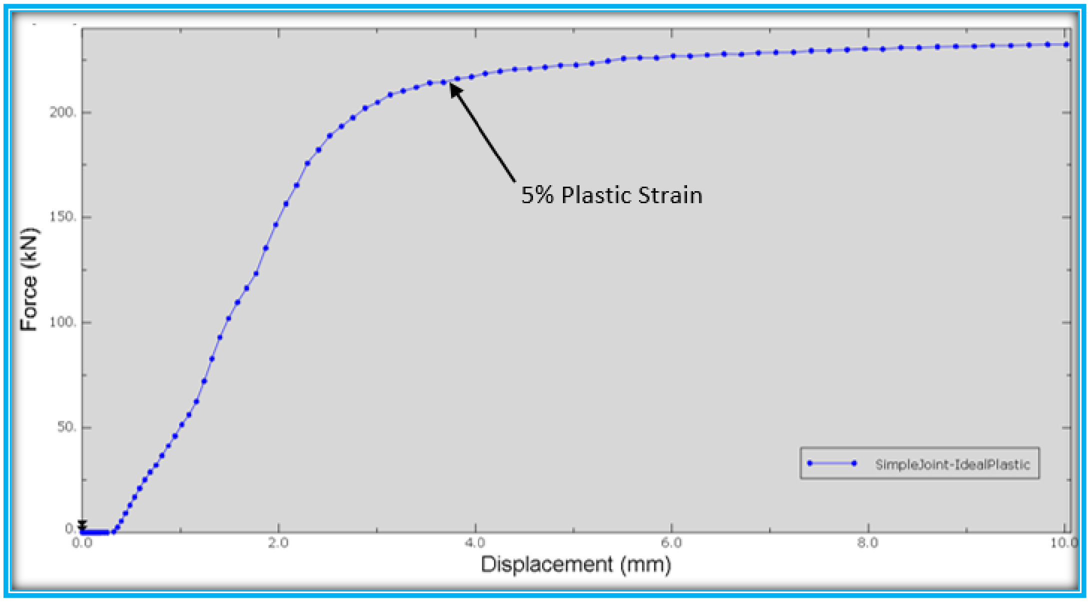


Figure 5- 8: Simple Joint - Force vs Displacement Curve (ABAQUS)

5.3.5) Comparison of Results

The comparison of the simple joint analysis have to be done directly between all of them. This means that the hand calculations without safety factors and the two numerical analyses have to be compared directly. The summary of the results is showed below:

Analysis	Model	Moment Resistance [kN*m]	Failure Mechanism
Handmade Calculation	With Partial Safety Factos	144.50	Bearing of the Beam Web
(EN 1993 1-8 2005)	Without Parcial Safety Factors	219.62	Bearing of the Beam Web
IDEA StatiCa	5% Plastic Strain Mechanism	97.00	Bending Plastification of the Column web
ABAQUS	Ideal Plastic Material	215.20	Bearing of the Fin Plate

Table 5- 11: Simple Joint - Analysis Results

Hand calculations have a good correlation with the numerical analysis results made by ABAQUS. There is only a difference of 2% between them. Nevertheless, the failure mechanism is different which indicates that the behavior is not the same for the two analyses. If the stress distribution is analyzed from ABAQUS, yielding of the beam web also happens. This suggests that both failure mechanism are in the joint simultaneously; however, bearing of the Fin Plate gave bigger plastic strains, which was the limitation parameter (see 5.2).

When IDEA StatiCa results are compered, a different failure mode is present, failure by bending of the column web. This failure mode cannot be predicted by the Eurocode guideline, because a stiff column face is assumed in the formulations. If we calculate a fin plate shear joint, there is no difference in the rules if one connects the fin plate to the column flange or the column web. However, the slenderness of the beam web needs to take a roll in the joint analysis.

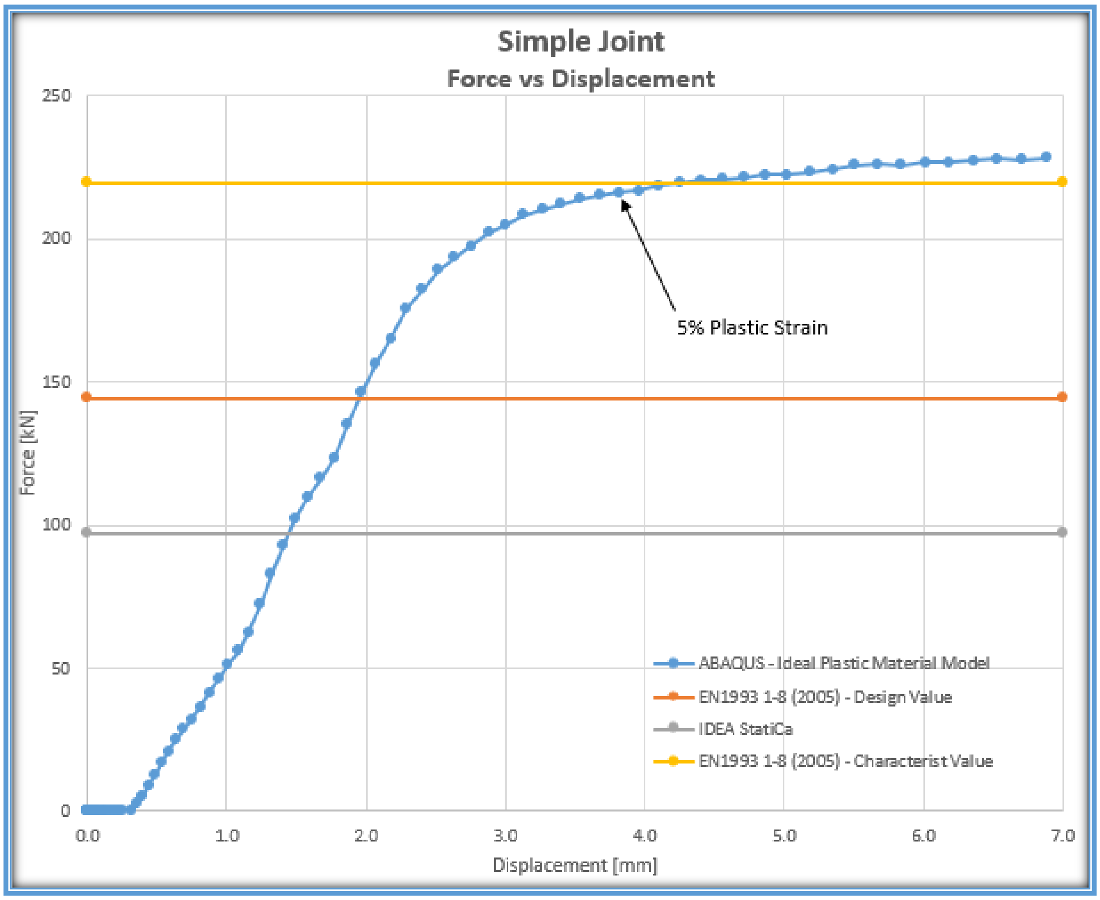


Figure 5- 9: Simple Joint Results

Bearing mechanism is a complicate failure. In most cases it comes along with the shear of the beam web or the fin plate. IDEA StatiCa evaluates the bearing capacity by using the formulation from the Eurocode; however, the forces are obtained from the finite element analysis. This means that bearing failure is linked to the bolt design but not to the yielding of the fin Plate (for the IDEA analysis).

Finally, because for this joint detail the hand calculation were compared only to the two numerical analyses, a mesh size study was performed in order to understand in better way the difference of the results between the two software. Figure 5-10 shows below the finite element size effect in both analyses.

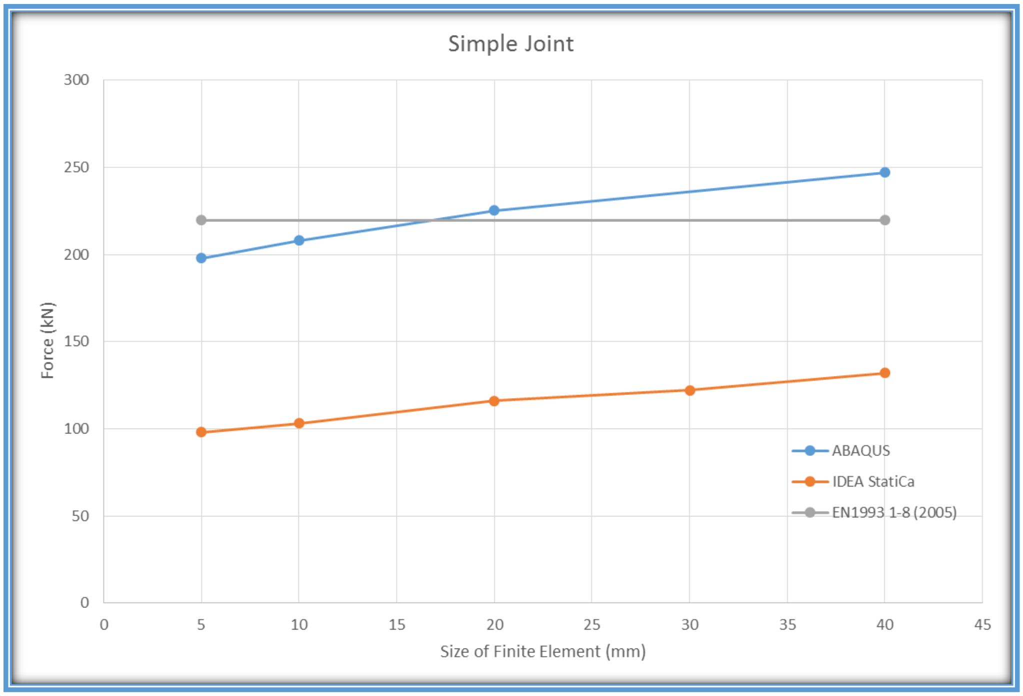


Figure 5- 10: Finite Element Size Effect

Both software shows the same tendency after the Finite Element Size effect study. The effect of "increase" of the joint strength when the mesh size is also increased, happened for both analyses in a similar proportion.

To gain more insight of the problem, an other study was performed. The failure of the column web was neglected for IDEA StatiCa calculations. When this parameter was applied, the joint strength was calculated in 213.5 [kN]. This value is closer to the results from ABAQUS model and the hand calculations. The IDEA StatiCa model overestimates the stresses concentrations that are producing the big plastic strains in the column web and determining the joint capacity.

5.4 Semi-Rigid Joint

5.4.1) Geometry Description

For the Semi-Rigid category, a bolted Flush End Plate Joint was chosen. This joint is part of the SERICOM database and the experiments were performed at TU Delft. The author of this investigation is Peter Zoetemeijer and it was performed in 1981. The detail is a one-side beam-to-column joint with flush end plate in a "T" configuration (see Figure 5- 11: Semi-Rigid Joint Details). The extremes of the column were supported (there is not information how was the boundary conditions at the column) and the beam was free (cantilever beam). The geometry and material properties are shown below:

Experiment T104-32

Element		Column	Beam	End Plate	Bolts	
Profile		HEM 450	IPE 400	PL386x180x10	M24 - 10.9	
Dimensions [mm]	*Length*	**2000.00	1350.00		24.00	Bolt diameter (tension) [mm]
	Heigth	478.00	400.00	386.00	20.00	Thickness of bolt nut [mm]
	Width	307.00	180.00	180.00	42.00	Across points dimension of bolt head & nut
	Flange Thickness	40.00	13.50	10.00	16.00	Thickness of bolt head [mm]
	Web Thicknes	21.00	8.60	-	4.00	Thickness of ALL washers [mm]
	Rolling Radius	27.00	21.00	-	44.00	Diameter of washer(s) [mm]
Material Properties [kN/mm²]	*Yielding Stress (fy)*	275.00	275.00	294.00	900.00	
	Ultimate Stress (fu)	*360.00	*360.00	*360.00	1000.00	
	Young Modulus	210000.00	210000.00	210000.00	*210000.00	

*Nominal Value

**Assumed Value

Table 5- 12: Semi-Rigid Joint Properties

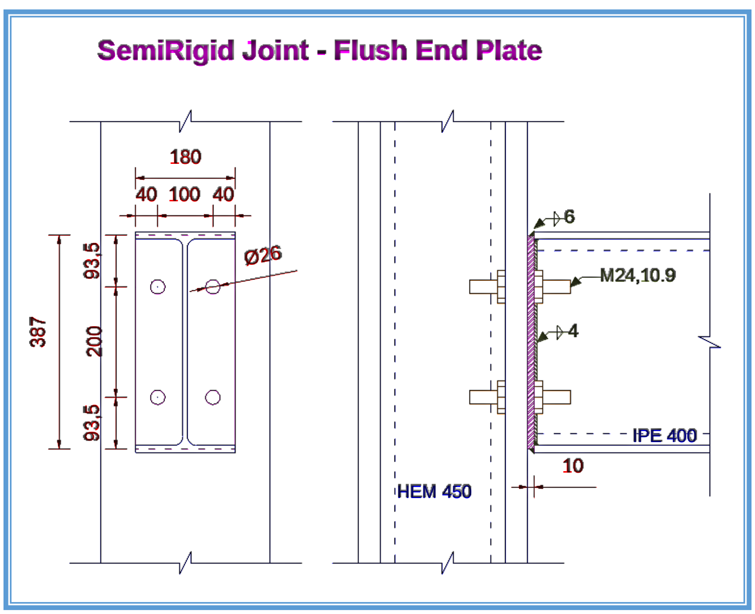


Figure 5- 11: Semi-Rigid Joint Details

5.4.2) Hand Calculation (Eurocode Rules 2005)

The hand calculations, using the Eurocode EN1993 1-8 (2005) give a joint resistance of 60.45 [kN*m] (see Annex C). This result is valid independent of the use of partial safety factors. This is because the weakest component in tension is the "End Plate in Bending" and this component has no safety factors in the T-Sub formulation. Mode 1 (plasticization of the plates, no contribution of the bolts) was obtained as the critical failure mode. The next failing component is tension in the beam web; nonetheless, the resistance is double of the bending End Plate resistance (see figure 5-4 and figure 5-5).

Failure Mode		Component Classification	EN 1993 1-8 (2005) [kN]	
			With Safety Factors	Without Safety Factors
F1.RD	Column Web Panel in Shear	Global Resistance	1711.87	1902.08
F2.RD	Column Web in Transverse Compression	Global Resistance	1603.51	2226.09
F3.I.RD	Column Web in Transverse Tension	Individual Tension Resistance	633.70	649.51
F3.G.RD	Column Web in Transverse Tension	Group Tension Resistance	1804.29	2310.00
F4.I.RD	Column Flange in Bending	Individual Tension Resistance	508.32	904.78
F4.G.RD	Column Flange in Bending	Group Tension Resistance	1016.64	1809.56
F5.I.RD	End Plate in Bending	Individual Tension Resistance	156.41	156.41
F5.G.RD	End Plate in Bending	Group Tension Resistance	436.68	303.25
F7.RD	Beam Flange & Web in Compression	Global Resistance	929.95	929.95
F8.I.RD	Bem Web in Tension	Individual Tension Resistance	503.81	503.81
F8.G.RD	Beam Web in Tension	Group Tension Resistance	976.81	976.81
F10.RD	Bolts in Tension	Global Resistance	1016.64	1809.56
F19.RD	Welds Resistance	Global Resistance	569.85	712.31

Table 5- 13: Semi-Rigid EN1993 1-8 (2005) Results

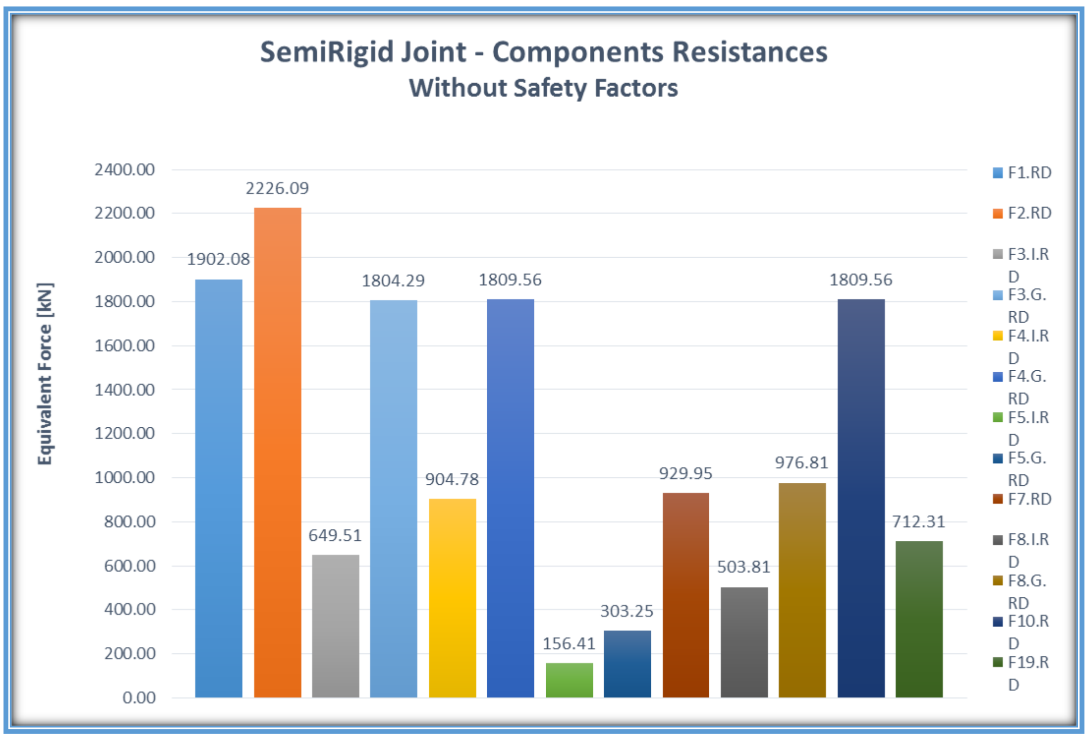


Figure 5- 12: Semi-Rigid EN 1993 1-8 (2005) Results

COP software was not possible to use, because of the joint geometry properties. The total width of the End Plate is not enough to have the minimum distances from the bolt to the edge and from the bolt to the column web. If the information from the SERICOM Database is used, then COP software does not allow the calculation of the Joint. The software needs a minimum of 185[mm] for the End Plate width, but it is only of 180[mm].

5.4.3) IDEA StatiCa Analysis

In the same way as the other joints, two values were calculated with IDEA StatiCa software. The first is the joint resistance using the Stiffness Analysis (see Annex A). Using the parameters proposed in 5.2, the calculated joint resistance is of 54.0[kN*m]. The second value was computed using the 5% plastic strain limit and the Stress/Strain Analysis. The joint resistance was of 62.50 [kN*m]. For both analyses, the failure mode can be found in the end plate. Figure 5- 13: Plastic Strain Distribution in the End Plate (IDEA StatiCa) shows yielding in most of the plate and the biggest plasticization happened in the tension part of the plate. By analyzing the deformation shape, the low utilization rate of the bolts and the plastic strain distribution, failure Mode 1 of the idealized T-Sub can be conclude to happen in the joint.

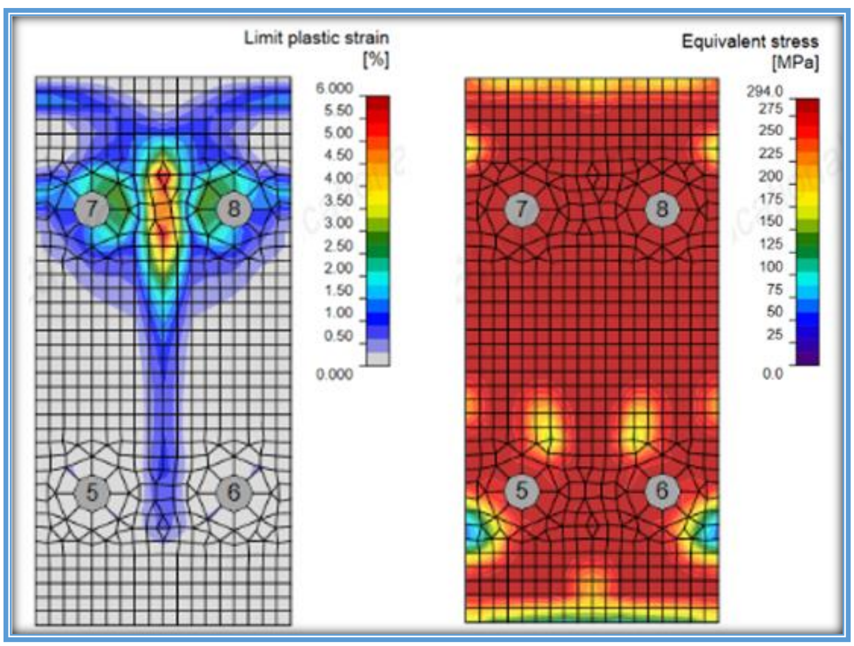


Figure 5- 13: Plastic Strain Distribution in the End Plate (IDEA StatiCa)

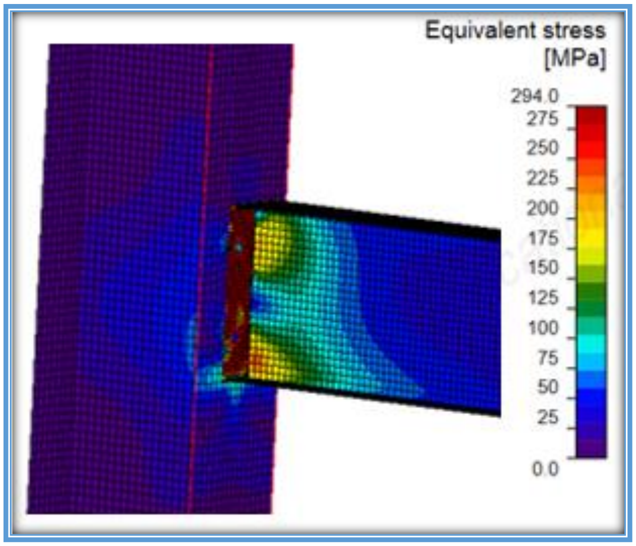


Figure 5- 14: Stress Distribution at the 5% Plastic Strain Limit

5.4.4) ABAQUS Analysis

In ABAQUS Software, the joint was modeled following the principles and assumptions stipulated in 5.2. Two material models were applied in the model. The first analysis was for an Ideal Plastic Steel Model. For the yielding stress the information from the database was used. This analysis gave an applied moment of 60.47 [kN*m] when the 5% of plastic strain was reached. Figure 5-7 shows the stress and strain distribution at the End Plate. It can bee seen that the failure mode is also happening at the end plate. Yielding of the element, the strain distribution and the deformed shape present a failure due to bending of the end plate. The Failure Mode 1 can be observed in Figure 5- 15: Plastic Strains & Stress Distribution at the End Plate (ABAQUS - Ideal Plastic Model).

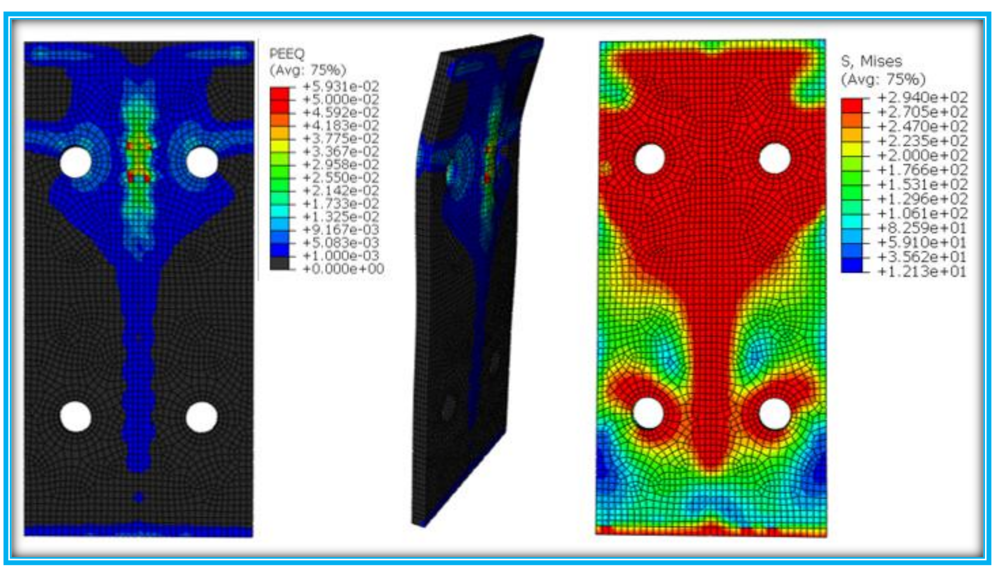


Figure 5- 15: Plastic Strains & Stress Distribution at the End Plate (ABAQUS - Ideal Plastic Model)

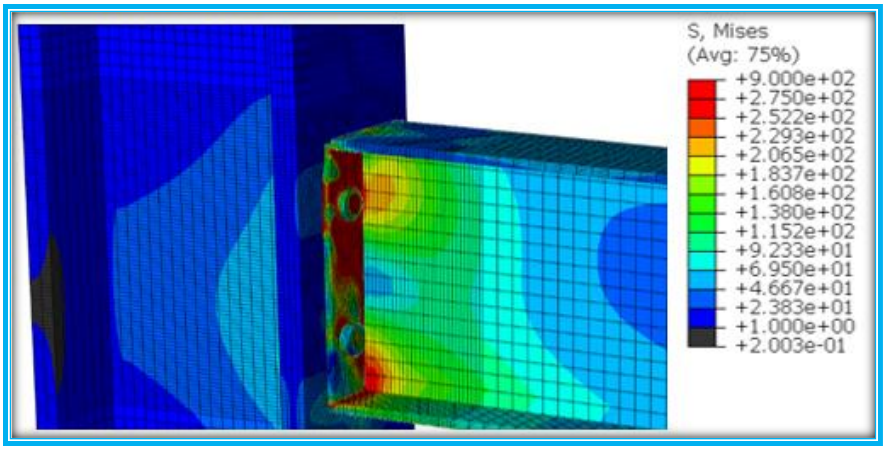


Figure 5- 16: Semi-Rigid Joint - Stress Distribution (ABAQUS - Ideal Plastic Model)

The second material model, Nominal Hardening gave a joint resistance of 98.11 [kN*m]. This value was computed when a 5% plastic strain mechanism could be found in the end plate. The failure mode is the same as in the first material model (End Plate in Bending). The Moment vs. Rotation curves for the two types of material are presented below:

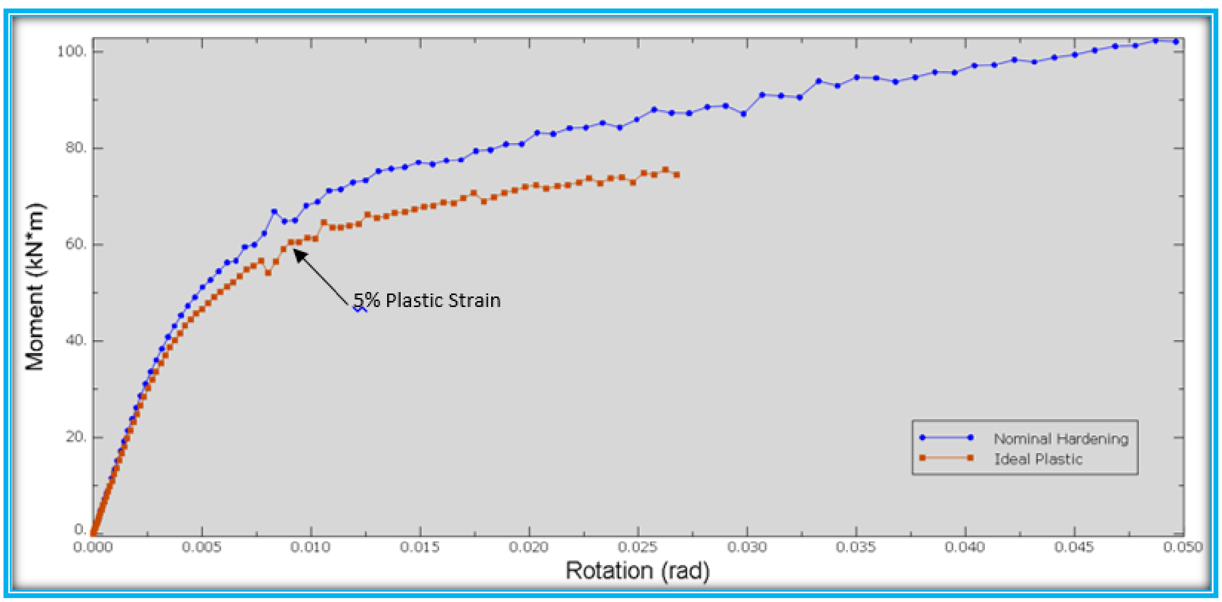


Figure 5- 17: Semi-Rigid Joint – Moment vs. Rotation Curves (ABAQUS)

5.4.5) Comparison of Results

The experiment results documented by Peter Zoetemeijer presented a joint resistance of 96.6 [kN*m]. The failure mechanism was not documented. However, there is a consistent result between all the analysis, failure mechanism of the joint is due to bending in the end plate. Furthermore, all the analysis show or describe a failure mechanism similar to the Failure Mode 1 of the T-Sub Idealization.

Even when the joint behavior and the deformed shapes are similar for the two numerical analysis, the results have some differences for the proposed limit. For the 5% plastic strain the accuracy is higher. The first comparison is between the hand calculations (EN 1993 1-8 (2005)) and the proposed limit for IDEA StatiCa. In this particular case, IDEA software predicts a joint resistance 10.67% smaller than the hand calculation. The predicted design joint resistance of IDEA StatiCa is in the safe side. However, the difference is close to 10% which is an acceptable difference.

The second comparison is between the two numerical models, ABAQUS with Ideal Plastic Material Model and IDEA StatiCa with 5% plastic strain, and the hand calculations for the characteristic joint resistance. ABAQUS numerical analysis predicts a joint resistance of 0.31% higher than the Eurocode rules. This is an perfect match in this detail. Nevertheless, the difference with IDEA StatiCa is only of 3.25%, which also is an accurate prediction for IDEA StatiCa calculations. This is a good validation for the ABAQUS Model and for IDEA results. Furthermore, the failure mechanisms are the same for the three analyses and the stress/strain distribution are similar for both Finite Element Analyses.

The last comparison is between the experiment results and the ABAQUS with Nominal Hardening Material Model. ABAQUS predicts a joint resistance 1.6% higher than the experiment final resistance. Both curves are matching in its behavior. This not only validates ABAQUS model but also gives a good insight of the whole analysis. The failure mechanism are the same. Nevertheless, there is a small difference in the initial stage of the joint behavior. For the experiment the elastic stiffness is higher than in the numerical analysis. This can be explain with the analysis of the bolt model. In this detail, the bolts are in tension, and the noise introduce to the behavior is due the not accuracy of the bolt model to replicate the real behavior of the bolt. In addition, this difference can also be explained by all the assumptions made in a "perfect model" which are used in ABAQUS Model. Nevertheless, the difference is so small that the accuracy of the prediction is high.

The summary of all the results are presented below:

Analysis	Model	Moment Resistance [kN*m]	Failure Mechanism
Experiment	***T106-004***	***96.60***	-
Handmade Calculation (EN 1993 1-8 2005)	With Partial Safety Factos	60.45	Bending of the End Plate (Mode 1)
	Without Parcial Safety Factors	60.45	Bending of the End Plate (Mode 1)
IDEA StatiCa	Propesed Limit	54.00	Bending of the End Plate
	5% Plastic Strain Mechanism	62.50	Bending of the End Plate
ABAQUS	Ideal Plastic Material (5% Plastic Strain)	62.47	Bending of the End Plate
	Nominal Hardening Material	98.11	Bending of the End Plate

Table 5- 14: Summary of Predicted Joint Resistances

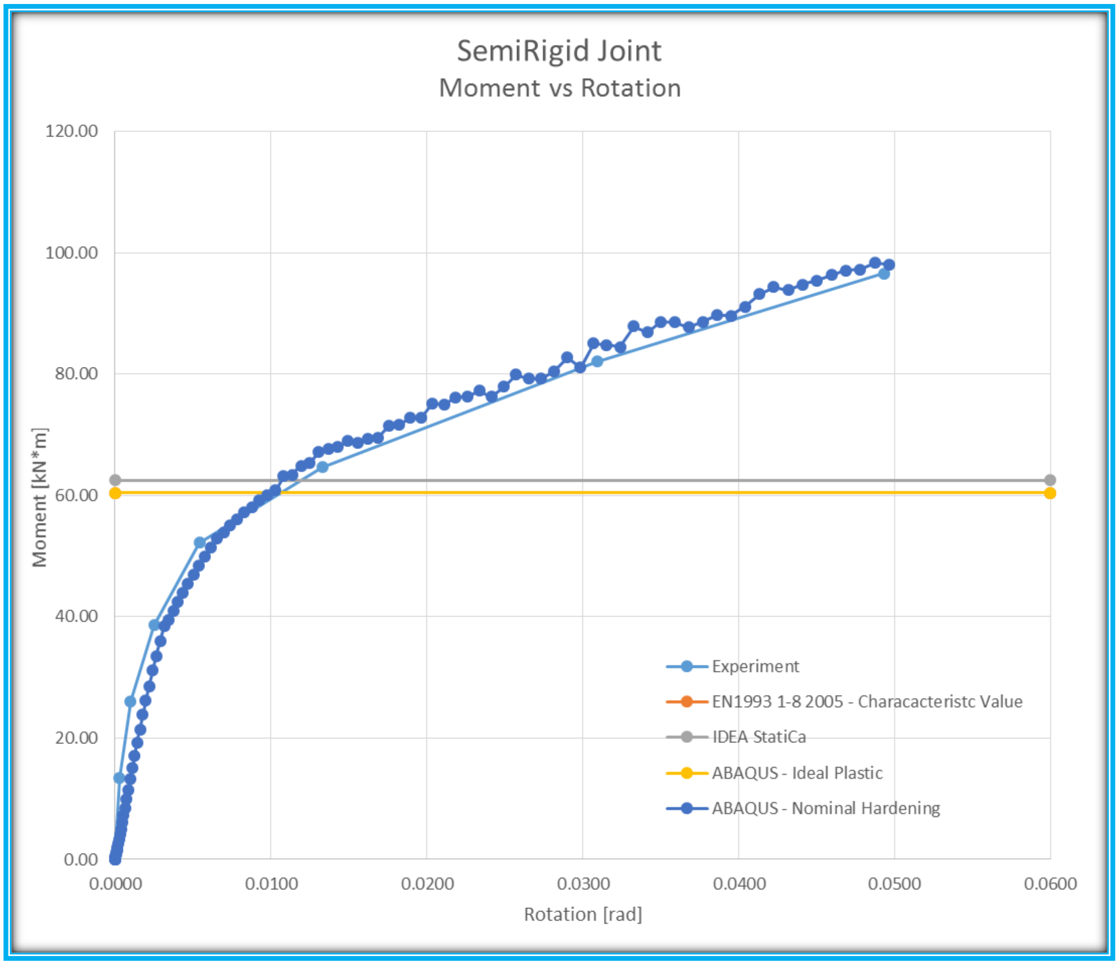


Figure 5- 18: Semi-Rigid Analysis Results

5.5 Rigid Joint

5.5.1) Geometry Description

For the Rigid Joint, a welded detail was selected in order to have variety in the types of steel joints. The Rigid Joint chosen corresponds to an experiment executed by Ch. Braun in 1987 at the Innsbruck University (Austria). The detail of the joint is a cantilever beam welded to the column flange, which has two supports making a T configuration. This experiment is documented in the SERICOM II database under the code of T106. The specific test that will be used for the comparison is number four. The full code is T106-004.

The geometry and detailing of the joint can be seen in Figure 5- 19: Rigid Joint Detail (T106-004). However, there is some missing information. The information from the SERICON II database specified that only the flanges of the beam are welded, no the web. The specified throat dimension is 14mm but there is no data regarding if this weld was made as a partial butt weld, as a single side filet weld or as double side filet weld. Nevertheless, welds are not the critical component of the joint. The critical component documented is the plasticization of the compressed zone of the column web. The assumption made was to use a double side fillet welds for the flanges of the beam. The properties of the profiles are presented below:

Experiment T106-004

Element		Column	Beam	Welds
Profile		HEM 200	IPE 500	a=14mm
Dimensions [mm]	*Length*	**1500.00	1150.00	14.00 Weld Throat
	Heigth	222.00	498.00	Double Side Fillet Weld
	Width	204.00	199.00	No Welds at the Beam Web
	Flange Thickness	25.60	15.10	Thickness of bolt head [mm]
	Web Thicknes	15.90	10.50	
	Rolling Radius	18.00	21.00	
Material Properties [kN/mm²]	*Yielding Stress (fy)*	267.00	248.00	248.00
	Ultimate Stress (fu)	*360.00	*360.00	360.00
	Young Modulus	210000.00	210000.00	*210000.00

*Nominal Value

**Assumed Value

Table 5- 15: Rigid Joint Profiles Properties

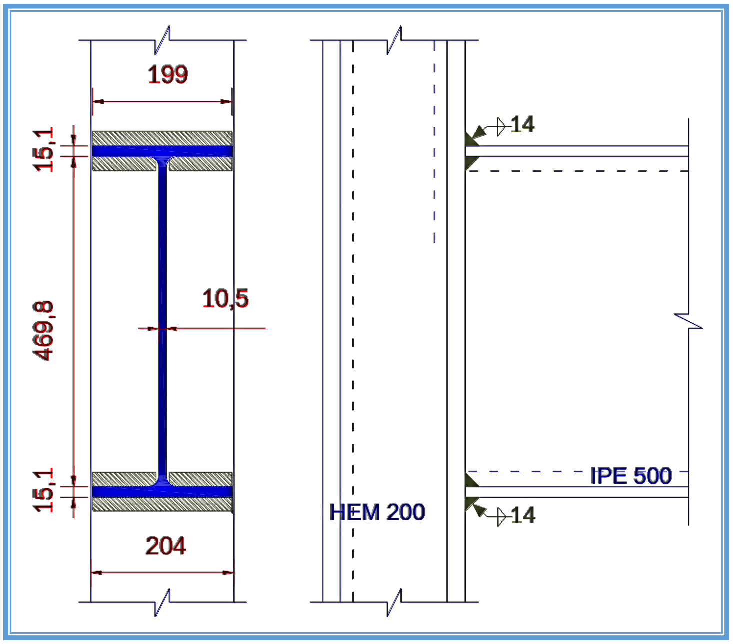


Figure 5- 19: Rigid Joint Detail (T106-004)

5.5.2) Hand Calculation (Eurocode Rules 2005)

Using the Eurocode EN1993 1-8 (2005), a joint resistance of 351.12 [kN*m] is calculated. The limiting component is the Column Web Panel in Shear (see Annex D). The next critical component is the Beam Flange/Web. However, the resistance of this component is 10.57% higher than the column flange in bending.

When the joint resistance is calculated with the Eurocode formulation but without considering the partial safety factors and the interaction factors (the characteristic value), a joint resistance of 359.86 [kN*m] is calculated. For this calculation, the limiting components are the Beam Flange in tension and compression. The next critical components are the Column Web in Shear and the Beam Web and Flange in tension. Nonetheless, they have a 8.50% more resistance.

The summary of the calculation of the different components is shown below:

Failure Mode		Component Classification	EN 1993 1-8 (2005) [kN]	
			With Safety Factors	Without Safety Factors
F1.RD	Column Web Panel in Shear	Global Resistance	727.69	808.55
F2.RD	Column Web in Transverse Compression	Global Resistance	840.62	1404.83
F3.RD	Column Web in Transverse Tension	Tension Resistance	924.68	1404.83
F4.RD	Column Flange in Bending	Tension Resistance	865.42	1579.61
F7.RD	Beam Flange/Web in Compression	Global Resistance	1126.76	745.22
F8.RD	Beam Flange/Web in Tension	Tension Resistance	804.60	745.22
F19.RD	Welds Resistance	Global Resistance	1094.49	1368.11

Table 5- 16: Rigid Joint - EN1993 1-8 (2005) Results

It can be seen in table 5-1, that the resistance of the components Beam Flange/Web in Compression and Tension are reducing their value when safety factors are taking away. This change happened because when the resistance of this component is calculated using the Eurocode rules, a global analysis of this component is performed. The Eurocode formulation does not contemplate that the web beam is not welded to the column face. Moreover, in this analysis, the plastic resistance for compression of the beam is used. In this case, the compression resistance is the sum of the flange and web under compression. However, this formulation does not take into consideration the local yielding of the column flange that will happen in this specific joint detailing. In the Rigid Joint, the beam is only connected through welds into the flanges, allowing the load to be transferred only by the flanges. The beam web should be neglected. By taking into consideration the specific situation of the joint, it can be see that the tension and the compression load of the flange are the same, and both components will be critical in the joint analysis. Tension and compression are the same for the beam flanges because the profile HEM200 has a cross section class 1. No buckling phenomenon is going to reduce the resistance due to compression of the flange plate.

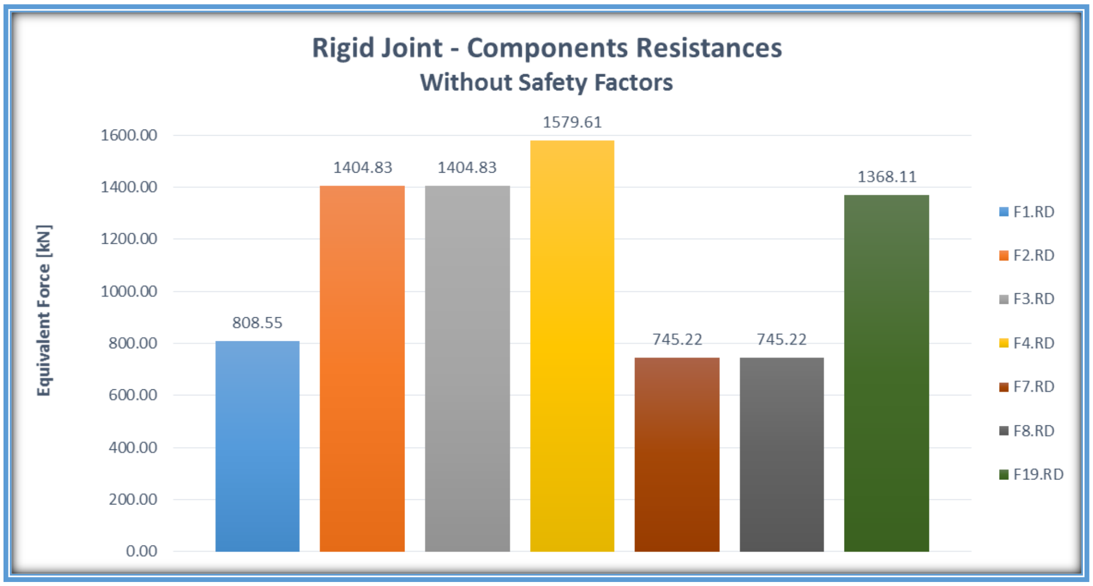


Figure 5- 20: Rigid Joint - EN1993 1-8 (2005) Results

The use of COP software for the verification of the hand calculations was not possible. First, the software does not allow only using welds at the flanges. The beam web welds are compulsory to use. Second, the software does not allow the modification of material properties. In SERICOM database, the different yielding stress for the flange and for the web can be found. These values came from the measurements during the experiments and not from nominal values, which are the values that COP uses in its calculations.

5.5.3) IDEA StatiCa Analysis

Two analysis where performed in IDEA StatiCa. First, using the Stiffness Analysis (see Annex A), the predicted joint capacity was of 423.2 [kN*m]. The biggest concentration of plastic strains can be found to happen at the beam flanges. A failure mechanism due to beam flange in tension or compression can be predicted. The second analysis was performed using the Stress/Strain Analysis. A limit of 5% of Plastic Strain was used to calculate the joint capacity. The calculated resistance was of 437.0 [kN*m]. The stress/strain distribution can be seen in Figure 5- 21: Rigid Joint - Stress/Strain Distribution (IDEA StatiCA), whose failure mechanism presented is the failure of the beam flanges.

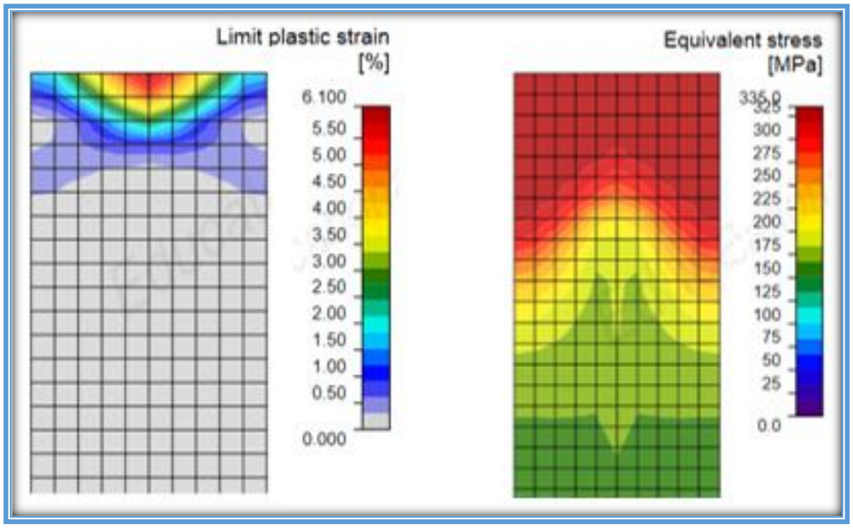


Figure 5- 21: Rigid Joint - Stress/Strain Distribution (IDEA StatiCA)

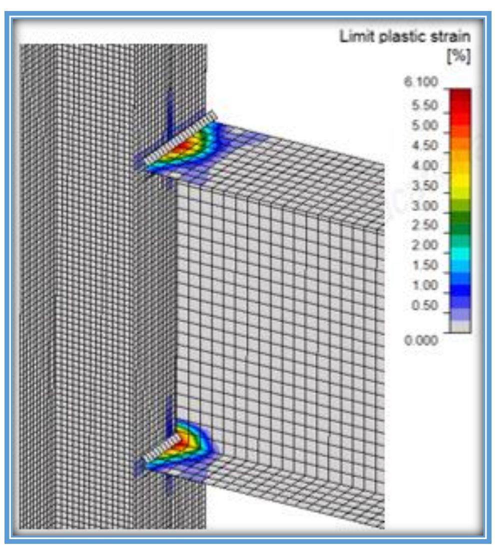


Figure 5- 22: Rigid Joint - Plastic Strain Distribution (IDEA StatiCa)

5.5.4) ABAQUS Analysis

Taking into consideration the modeling assumptions, simplifications and parameters for the ABAQUS analysis (see 5.2), a joint resistance of 361.68 [kN*m] is calculated for the Ideal Plastic Material Model. The 5% Plastic Strain mechanism happened in both flanges at the same time. This can be expected due to the local transfer of the load (only through the flanges welds) and because the IPE 500 is a class 1 profile. Class 1 cross-sections can reach the plastic resistance when compression is applied to them. This is why the tension flange and the compression flange have the same resistance and they fail simultaneously.

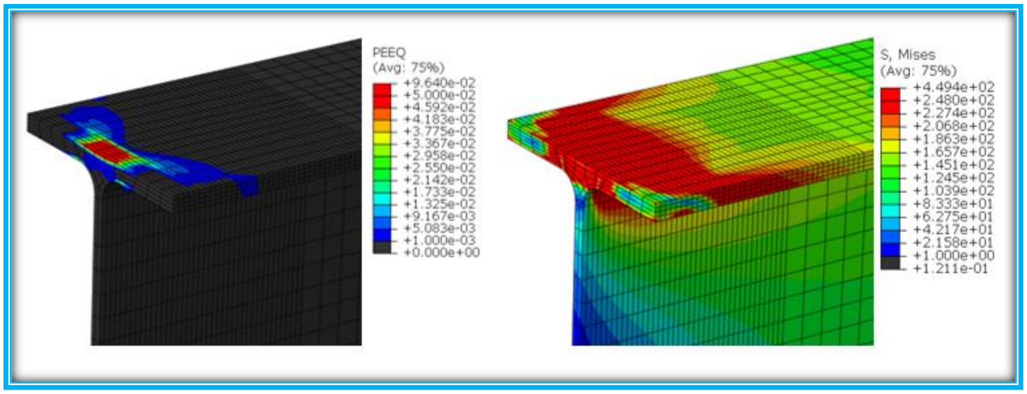


Figure 5- 23: Rigid Joint - Strain and Stress Distribution of the Beam Flange (ABAQUS - Ideal Plastic)

Figure 5- 24: Rigid Joint – Strain/Stress Distribution (ABAQUS - Ideal Plastic) shows the stress distribution in the entire joint. The column web is yielding, however the plastic strain in it is considerably small. For this reason, the failing mechanism selected is the plasticization of the beam flanges.

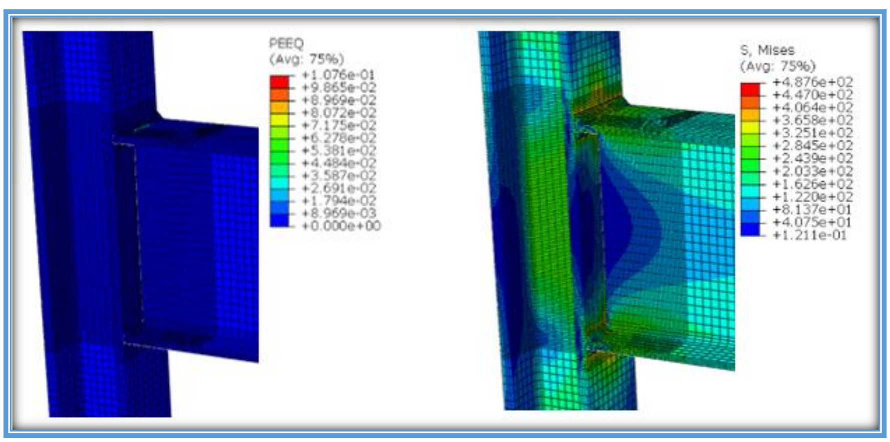


Figure 5- 24: Rigid Joint – Strain/Stress Distribution (ABAQUS - Ideal Plastic)

In the second ABAQUS analysis, where the Nominal Hardening Material Model was used, a joint resistant of 431.71 [kN*m] was calculated when a mechanism of 5% Plastic strain happen. In the same way as the Ideal Plastic Material Model, the failing mechanism is the plasticization of the beam flanges. The Moment vs Rotation curves for both material models are shown in Figure 5- 25: Rigid Joint – Moment vs Rotation Curves (ABAQUS).

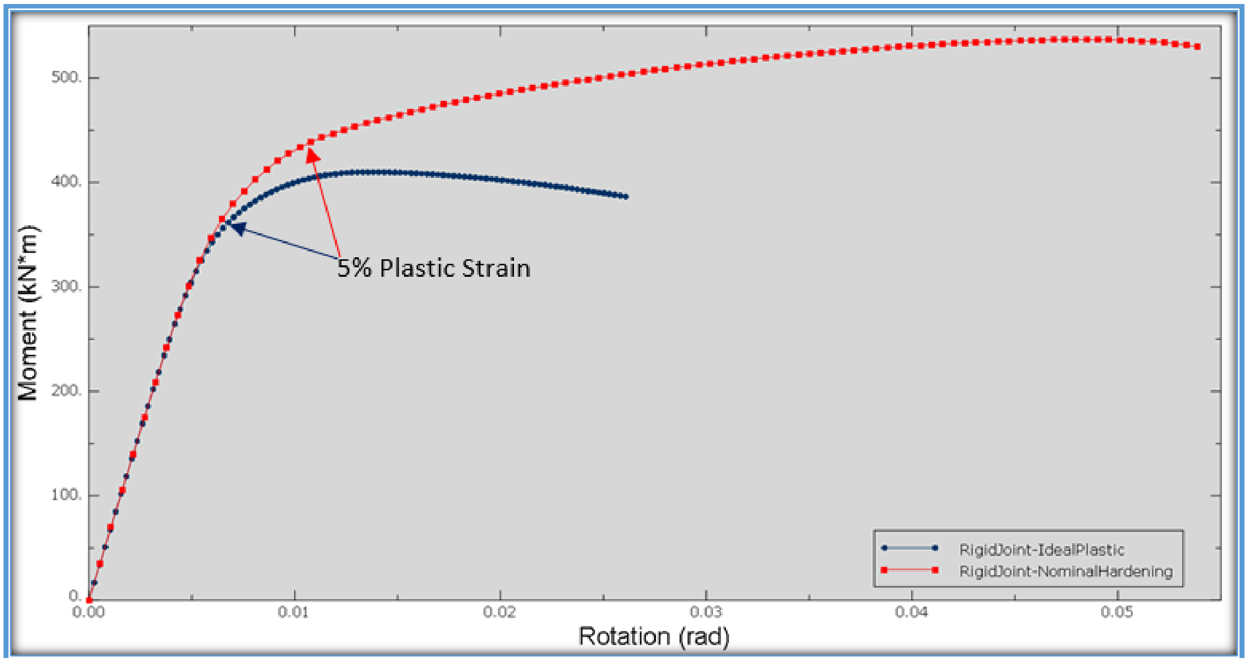


Figure 5- 25: Rigid Joint – Moment vs Rotation Curves (ABAQUS)

5.5.5) Comparison of Results

In SERICOM database, the failure mode documented for the Test T106-004 performed by Ch. Braun, is the plasticization of the compressed column web with a maximum moment resistance of 433.69 [kN*m]. The location of the failure mechanism is close to the failure mechanism obtained in the different analysis; however, the failure mechanism predicted is the tension and compression failure of the beam flanges. The column web reached its yielding stress and high concentration stress can be found in the contact zone between the beam flange and the column face. This can lead to both, a misinterpretation of the results or that the missing data SERICOM is playing a big role. Nevertheless, the joint capacity is congruent with the predicted values. The summary of the results is shown below:

Analysis	Model	Moment Resistance [kN*m]	Failure Mechanism
Experiment	***T106-004***	***433.69***	***Plastification of the Compressed Column Web***
Handmade Calculation (EN 1993 1-8 2005)	With Partial Safety Factos	351.12	Column Web in Shear
	Without Parcial Safety Factors	359.86	Beam Flanges in Tension and Compression
IDEA StatiCa	Propesed Limit	423.20	Beam Flanges in Tension and Compression
	5% Plastic Strain Mechanism	437.00	Beam Flanges in Tension and Compression
ABAQUS	Ideal Plastic Material	361.68	Beam Flanges in Tension and Compression
	Nominal Hardening Material	431.71	Beam Flanges in Tension and Compression

Table 5- 17: Rigid Joint Results

Table 5-2 shows the different results from all the analysis; however, the comparison should be made in the following way:

- Hand calculations, using the Eurocode rules without safety factors or interaction factors, should be compared to the ABAQUS numerical analysis using the Ideal Plastic Material Model and the IDEA StatiCa 5% limit Plastic Strain analysis. The difference between ABAQUS and hand calculations is 0.5%. These results validate the modeling technique in ABAQUS and the calculations based in the Eurocode. However, IDEA StatiCa predicts a joint resistance 21.43% higher than the other two analysis. This numerical analysis gave unsafe results. Nevertheless, all the analyses predict the same failure mechanism. The stress/strain distributions are also similar for the two numerical analyses.
- Hand Calculation of EN1993 1-8 (2005) using the safety factors should be compared to the proposed limit for IDEA StatiCa. In this joint detail, IDEA predicts 20.5% higher resistance. This is an unsafe result. Using a plastic limit strain of 2% gave a difference only of 8.1%. The conclusion that emerges from this comparison is that when a failure mechanism depends on a brittle element like bolts and in this case the welds, the proposed limit using 3% of plastic strain should be reduced to 2% (389.0 [kN*m] is the calculated joint resistance using 2% plastic strain) .
- The experiment behavior and results should be compared against the ABAQUS numerical model using Nominal Hardening Material Model. Using the limit of 5% plastic strain mechanism, the joint resistance has a difference of 1.4% higher for the numerical analysis. Nevertheless, the behavior is quite different. The experiment curve is considerably stiffer at the elastic stage than the numerical analysis. This can be explained due to the stiffness used for the welds. In the numerical model, the welds were modeled using the same stiffness than the steel. In reality, welds are stiff elements with low deformation capacity; consequently, the plastic strain cannot match between the welds and the beam flanges plates. In the inelastic stage, the curves are similar (see Figure 5- 26: Rigid Joint Results. Additionally, at the 5% limit stage, the experimental joint behavior is close to the numerical joint behavior. These results give a respectable validation of the modeling technique and the predicted results using the different analysis.

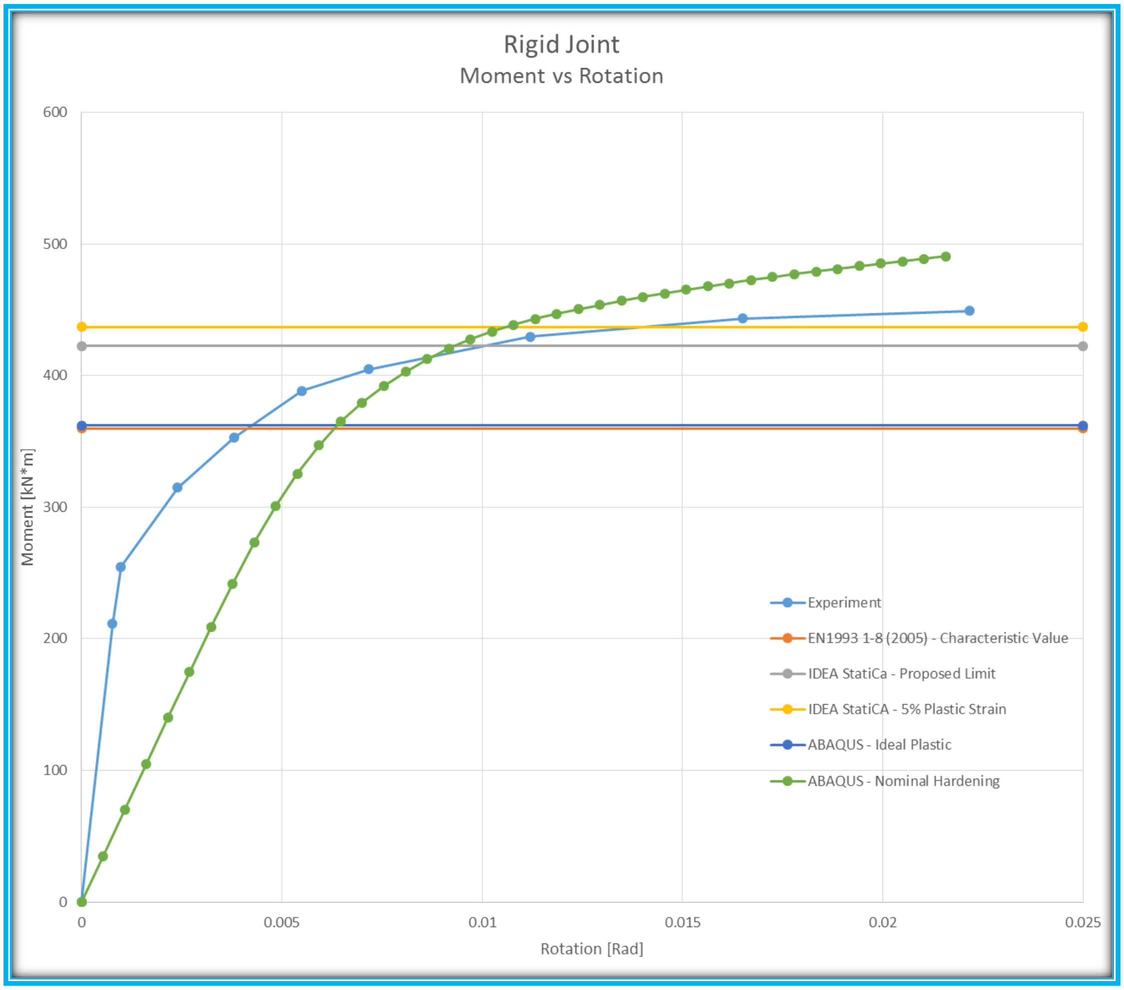


Figure 5- 26: Rigid Joint Results

Chapter 6 – Conclusions and Recommendations

6.1 Conclusions

6.1.1) Chapter 3 – Experiment

- The failure value of the hybrid weld of the first specimen (1E1) was 743.20 [kN]. This weld resistance is 45% bigger than the calculated resistance (512.50 kN) using draft EN1993 1-8 (2020). The Eurocode prediction is rather conservative. If the column face is assumed infinitely stiff, the hand calculated resistance of the weld under pure shear is 723kN. This value becomes closer to the experimentally obtained value.
- The higher real resistance of the weld compared to the new formulation of the code validate the safety and reliability of them. This allows to take into consideration the filler material properties, which will permit to predict more accurate values for the weld resistance, in relation to the actual formulation in EN1993 1-8 (2005), which is 40% smaller than the experiment .
- The higher value obtained in the experiment compared to the predicted by following the Eurocode draft (EN1993 1-8 (2020)), validates the use of this specific detail in engineer practice. This type of simple joint is common used by the steel industry and by ASK Romein.
- The resistance prediction made by IDEA StatiCa was 1.66% higher than the Eurocode (2005) prediction. Taking into consideration that the design value that is legally accepted is the one obtained from the Eurocode formulation, IDEA StatiCa can be use in daily practice for this type of joint detail and failure mode. No finite element size effect were found during the analysis. The joint strength was reduced by 4% when the size of the finite element was reduced from 30mm to 5mm.
- The idealization of a hinge for this type of simple joints is correct. The expected spring behavior at the face of the column was developed. The flexibility of the column face produced a reduction of the bending moments at welds. The actual load transfer to the weld is close to pure shear loading. The distribution of moments along the beam was linear during the elastic stage, which also support the assumption of a nominally pinned joint. This behavior was also found in the numerical simulation made in ABAQUS.
- The rotation center of the experiment joint during the elastic stage was found to be 10mm closer to the column face than the rotation center assumed by the code and the guidelines. This lead to smaller bending moment at the weld (10% smaller), validating the code assumption to be in the safe side but with enough accuracy.
- The rotation center of the experiment joint during the plastic stage was found to be 37mm closer to the column face than the rotation center assumed by the code. This shift of the rotation center during the plastic stage had a beneficial effect in the weld. The bending moments in the weld were decreased and the stress distribution at the weld was changed. During the elastic stage, the presence of perpendicular normal stresses ($\sigma_{\perp}$) and both shear

stresses (parallel $\tau_{\parallel}$ and perpendicular $\tau_{\perp}$) can be found at the weld throat (a) cross-section. When the plastic stage is reached, the perpendicular normal stresses ($\sigma_{\perp}$) and the shear stresses ($\tau_{\perp}$) are considerably reduce and only the parallel shear stress remain dominant at the throat cross-section.

- The shift of the rotation center at the plastic stage also changed the load distribution in the bolts by placing more load in the first column of bolts. Most of the bearing deformation of the Fin Plate and the Beam web happened in this bolt row.
- The numerical model using nominal hardening material model was able to predict in an accurate way the experimental behavior of the joint. Not only it predicts in a precise manner the elastic stage, but also the nonlinear stage was replicated. The stiffness of the joint in the experiment during the first cycle load was possible to simulate. In addition, the elastic limit for the coming load cycles were increased because of the yield flow due to the hardening in the joint after the first cycle load. The variation of the elastic limit of the joint was accurate reproduced by the numerical model.
- The proposed boundary conditions for the experiment were able to generate the two rotation points (at the column face and the fin plate) in the joint. In this way, the Eurocode calculation model was possible to validated and the real behavior of the joint in common practice replicated.

6.1.2) Chapter 4 – Hybrid Welds

- Using the filler material properties, which are present in the new procedure to obtain fillet welds strength in the draft version of EN1993 1-8 (2020) gives an increase in the weld strength of 22%, using steel plates with S355 and the filler wire MC715-H (f_u=580N/mm^2). These properties for the filler material are commonly used by the Steel Industry in the Netherlands.
- The simplified finite model for welds, which used the Von Mises yielding criterion gave 7% lower resistance for tension loading and 23% lower resistance for shear loading than the calculated by the new EN1993 1-8. This might suggests that the Von Mises criterion is not accurate when describing the real behavior of welds.
- For shear loaded welds, a weld size effect can be seen for the numerical model. For welds with length smaller than 50 times the throat thickness (a), the numerical model will underestimate the weld resistance due to size effect.

6.1.3) Chapter 5 – Joint Design Assessment

ABAQUS

- The 3D model with Ideal Plastic material used in ABQUS, which can be replicated in any general purpose finite element software predicts in the three cases a joint strength with a difference less than 5% compared to the characteristic resistance calculated with EN1993 1-8 (2005).
- The failure mechanisms obtained in the numerical analysis in ABAQUS matches with the failure mechanism obtained by the hand calculations in the analyzed joints. Having an accurate strength prediction and the same behavior allows validating the model technique used in ABAQUS.

- Using Nominal Hardening material model stipulated in EN1993 1-5 in the same proposed model, gives a good matching in the behavior and strength of the steel joints compared to the documented experiments. In the case of the semi-rigid joint, a difference less than 1% was found. For the experiment and the rigid joint a difference of 10% was obtained. This value is acceptable because of the nominal properties used in the models. If the predicted strength using the Nominal Hardening Material Model is compared to the characteristic resistance calculated with the code (EN1993 1-8 (2005)), a difference of 23% is obtained against the rigid joint, 34% against the experimental joint and 60% against the semi-rigid joint.

IDEA StatiCa

- IDEA StatiCa is suitable software for daily practice. Its good predictions of joint strength and behavior, plus the friendly interface and set up, makes IDEA StatiCa appropriated for steel joints design. Nevertheless, the validation process needs to be extended to more joint types and with increased complexity of joint detail.
- For the Semi-Rigid (Flush End Plate) joint, the strength, failure mode, stress and strain distribution, matches to the ABAQUS predictions and hand calculations with great accuracy. The "T-sub" idealization, specifically its failure mode 1 (yielding of the plate) can be accurately modeled and replicated by the software. It was found a difference of 3% compared to the hand calculations and less than 1% compared to the ABAQUS model with ideal plastic material model.
- Using the 5% Plastic strain proposed by the software and Annex C of EN1993 1-5, gives accurate results compared to the characteristic value. In the three joints, for the ABAQUS model, the 5% plastic strain limit was presented in the transition zone of the Force vs Displacement. For this reason, it is suitable to use this strain value as the failure criterion to predict the joint resistance.
- Using the plastic material model in IDEA StatiCa for modeling welds, gives similar results as the strength calculated with the new formulation of EN1993 1-8 (2020), leading to a difference less than 2%.
- In the simple joint, where the joint configuration was in the weak axis (attached to the beam web), IDEA StatiCa predicts a plasticization pattern at the extremes of the weld. The higher concentration at the end of the welds produce high plastic strain at the column web, which is limiting the joint capacity according to the software. For these reasons IDEA StatiCa underestimated the column web resistance almost with a factor of 2 compared to the ABAQUS model and the hand calculations. However, if this failure mechanism is neglected, the joint strength is similar to the ones from the other analyses and the stress distribution in the web of the "I" shape column, is similar for both numerical software. Special attention in result interpretation is needed when this type of details are analyzed by IDEA StatiCa.

6.2 Recommendations

General recommendations for the continuity of the experimental program and the validation of IDEA StatiCa are presented below:

- The numerical model to predict the experiment behavior needs to be improved by using a true stress / true strain material model of the weld material. Coupon tensile tests need to be performed.
- The numerical model can be improved by applying a damage control material in order to predict the failure and crack propagation of the weld due to the concentration stress at the end of it. The prediction of the appearance of the first crack in the numerical model needs to be validated by using a camera in the experimental tests. If the initial slip of the bolts want to be simulated by the numerical model, a preload force equal to the applied one in the experiment (30N*m), should be applied to the bolts.
- The deformation of the hollow section face was evident during the test. By analyzing the results, it can be concluded that the contribution of the column face to the moment resistance is small. Nevertheless a “basic component” to predict the behavior of the hollow section column face is needed in order to improve the calculation model of simple joints.
- To have a proper frame design, the maximum load expected during the experiments needs to be calculated by using a nominal hardening material or taking and extra 10% if the load is calculated using an elastic approach. This load should be multiplied by a factor of 3 and then use it as the design load of the experiment frame. The frame needs to be design avoiding any plastic strain happening on it.
- The design of weld resistance by using the Eurocode rules leads to a conservative result of the weld strength. More experiments, especially using real size joints, need to be performed in order to reduce scattering as it is observed in the experiment performed.
- A study about the use of the Von Mises criteria for the analysis of welds might be worth of further investigation.
- The normal (use for S355 elements) filler material had a good performance in the joint. If the HSS weld is able to replicate the performance, then important savings in weld costs can be made.

Bibliography

Bayo, E., Cabrero, J. M., & Gil, B. (2006). An effective component-based method to model semi-rigid connections for the global analysis of steel and composite structures, *28*, 97–108. https://doi.org/10.1016/j.engstruct.2005.08.001

Beg, D., Zupančič, E., & Vayas, I. (2004). On the rotation capacity of moment connections. *Journal of Constructional Steel Research*, *60*(3–5), 601–620. https://doi.org/10.1016/S0143-974X(03)00132-9

Bijlaard, F. (2006). Eurocode 3, a basis for further development in joint design. *Journal of Constructional Steel Research*, *62*(11), 1060–1067. https://doi.org/10.1016/j.jcsr.2006.06.012

Bridge, L., & Specification, D. (2011). 2 . Design of Welded Connections. *Society*.

Bursi, O. S., & Jaspart, J. P. (1997). Benchmarks for Finite Element Modelling of Bolted Steel Connections. *J. Construct. Steel Res*, *43*(3), 17–42. https://doi.org/10.1016/S0143-974X(97)00031-X

Bursi, O. S., & Jaspart, J. P. (1997). Calibration of a Finite Element Model for Isolated Bolted End-Plate Steel Connections. *J. Construct. Steel Res*, *44*(3), 225–262. https://doi.org/10.1016/S0143-974X(97)00056-4

Dvořáková, E., & Patzák, B. (2016). Four-Node Quadrilateral Shell Element MITC4. *Applied Mechanics and Materials*, *825*(2), 99–104. https://doi.org/10.4028/www.scientific.net/AMM.825.99

Dvorkin E Bathe K. (1984). A continuum mechanics based four-node shell element for general non- linear analysis. *Engineering Computations*, *1*, 77–88. Retrieved from http://www.simytec.com.ar/docs/Dvorkin_Bathe.pdf

Eurocode 3 1-8 Comitte. (2005). EN1993: Eurocode 3 Design of steel structures - Part 1-8: Design of joints (2005). *Eurocode 3*, *8*(2005), 135.

Eurocode3 1-5 Comitte. (2006). EN1993: Eurocode 3 Design of steel structures - Part 1-5: General rules - Plated structural elements. *Eurocode 3*, *5*(2006).

Eurocode3 1-8 Comitte. (2017). EN1993: Eurocode 3 Design of steel structures - Part 1-8 : Design of joints (2020) - Draft 2.1, *2005*(May), 1–181.

H.P. Günther, J. Hildebrand, C. Rasche, C. Versch, I. Wudtke, U. Kuhlmann, M. Vormwald, F. W. (2009). Welded connections of high strength steels for the building industry, (March), 1–41.

Ideas, P., & Professional, D. (1997). Use Undermatching Weld Metal Where Advantageous, *XIV*(1).

Jaspart, J.-P., & Weynand, K. (2016). *Design of Joints in Steel and Composite Structures*. Mem Martins: ECCS - European Convention for Constructional Steelwork.

Jiménez de Cisneros, J. J., Carazo Álvarez, D., Carazo Álvarez, J. D., Jordão, S., & Simões da Silva, L. (2017). Análisis de componentes T-equivalente asimétricas: enfoque experimental y numérico. *Revista Internacional de Métodos Numéricos Para Cálculo Y Diseño En Ingeniería*, *33*(1–2), 129–137. https://doi.org/10.1016/j.rimni.2016.04.001

Kim, J., Yoon, J.-C., & Kang, B.-S. (2007). Finite element analysis and modeling of structure with bolted joints. *Applied Mathematical Modelling*, *31*(5), 895–911. https://doi.org/10.1016/j.apm.2006.03.020

Mago, N., & Hicks, S. (2012). Finite Element Analysis: What it is and how can it help your company? *Innovation in Metals*, (C), 1–3. Retrieved from https://www.hera.org.nz/Folder?Action=View File&Folder_id=89&File=What is FEA and how can it help you.pdf

Rasche, C., & Kuhlmann, U. (2012). For determining the load - bearing capacity of Fillet welds of higher structural steels. *Stahlbau*, *81*(11), 889–897. https://doi.org/10.1002/stab.201201628

Simões da Silva, L. (2008). Towards a consistent design approach for steel joints under generalized loading. *Journal of Constructional Steel Research*, *64*(9), 1059–1075. https://doi.org/10.1016/j.jcsr.2008.02.017

Simões da Silva, L. (n.d.). SERICOM II - Database. Steel Joints European Database.

Simões da Silva, L., Lima, L., Vellasco, P., & Andrade, S. (2002). Experimental behaviour of end-plate beam-to-column joints under bending and axial force, 1–15.

Simoes da Silva, L., & Girao, A. (2001). An analytical evaluation of the response of steel joints under bending and axial force. *Computers & Structures*, *79*, 873–881. https://doi.org/10.1016/S0045-7949(00)00179-6

Törnblom, S. (2007). Undermatching Butt Welds in High Strength Steel.

Williams, J. G., Anley, R. E., Nash, D. H., & Gray, T. G. F. (2009). Analysis of externally loaded bolted joints: Analytical, computational and experimental study. *International Journal of Pressure Vessels and Piping*, *86*(7), 420–427. https://doi.org/10.1016/j.ijpvp.2009.01.006

CARACTERIZAÇÃO E SISTEMATIZAÇÃO DO COMPORTAMENTO EXPERIMENTAL DE. (2002).

Annex A – Setup Guideline for IDEA StatiCa

A.1 Introduction

IDEA StatiCa, like any other program that uses finite element analysis of structures or elements, requires an initial configuration in order to obtain reliable, safe, practical and accurate results. In this annex, a proposal is made for the initial configuration of IDEA StatiCa based on the parametric study carried out in chapter 5 of this thesis. In addition, the information provided by the developers of the program was taken into account and it was analyzed. The information was gathered from the company website (IDEA StatiCa, 2017) and the book "Benchmark Cases for Advance Design of Structural Steel Connections" (František, 2016) where the analysis of different cases studies are made using IDEA StatiCa. This guideline also presents the results of different parametric studies carried out to determine important parameters in the analysis of steel joints, such as finite element size.

IDEA StatiCa software contains three packages: steel joints, steel beam design and pre-stressed concrete beam design. In this guide, the steel joint designer package will be described since it was used throughout the master project.

Figure A - 1: Initial Page of IDEA StatiCa

A.2 General Overview

The program developers describe IDEA StatiCa as follows: "Engineering software dedicated to structural design and code-check joints, cross sections, beams and other details" (IDEA StatiCa, 2017). The principles and parameters that define the way that IDEA StatiCa works will be presented below.

A.2.1) Description

IDEA StatiCa is a program specialized in the design and analysis of structural details. It combines finite element analysis with the analytical method of steel design based on the component method (CM) regulated in Eurocode 1993-1.8 (see chapter 2). The two design approaches are combined. The results obtained by the FEA are checked with the design rules established in two codes: the European code and the US code for structural steel design (AISC).

Being a program focused on the local design of details, its tools and interface are dedicated to the modeling of steel joints. This means that IDEA StatiCa has a set of tools and options that facilitate the modeling and analysis. This allows a standard joint to be analyzed within a few minutes, obtaining results comparable to those obtained with other finite element programs (see chapter 5).For this reason, the software can be used in the daily practice of steel joints design.

IDEA StatiCa performs two types of analysis:

1) A geometrically linear analysis with non-linear properties of materials and contact constrains, to find stresses and strains.
2) A linear buckling analysis (eigenvalue analysis) where the critical buckling load factor is obtained. This factor is used to perform the joint stability check.

A.2.2) CEBFM

The way IDEA StatiCa combines FEA and design analytical rules is through its so-called "Component Base Finite Element Model (CBFEM)" (IDEA StatiCa, 2017). This method uses the component method specified in the Eurocode, but for the analysis of the stresses in the individual components, it uses a finite element analysis. A clear advantage of this is that the method is not limited by the standard joints specified in the code, but can be extended to complex (non-standard) joint analysis. Eurocode 1993 1-8 has regulated only components for joints between elements of "I" shape cross-sections.

One of the challenges of the component method is the assembly of springs (both in parallel and in series), which requires a more complex analysis to be able to determine the behavior and capacity of the joint. The assembly of spring's model for joints that do not have a simple geometry can become complicated and demands much attention and experience. In the case of the CBFEM method, the assembly of the spring's model is implicitly executed when the finite element analysis is performed.

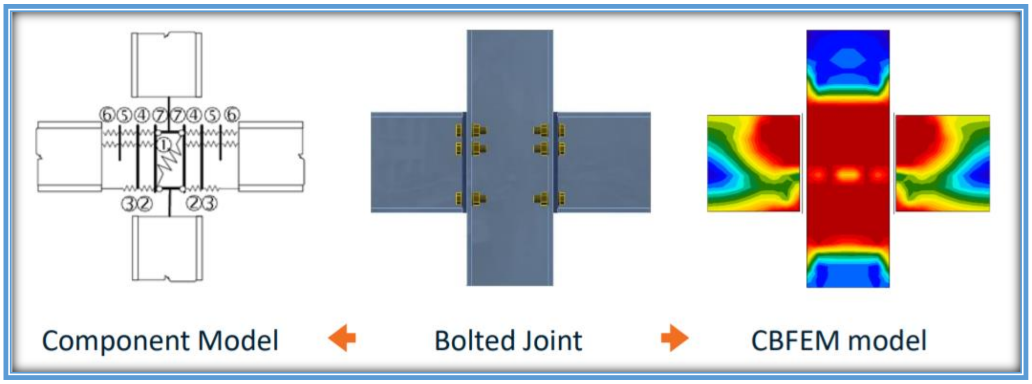


Figure A - 2: Component Model vs CBFEM Model

A.2.3) Material Model

To keep the analysis simple and fast, IDEA StatiCa uses a simplified material model. Although it is a non-linear material (is not infinite elastic) and has a complex hardening stage, steel is modeled as a bilinear material. This means that the material will behave with some rigidity (Young Modulus) to a specified yielding point, and from that point the material will deform infinitely without increasing the stresses. The hardening behavior of the steel is not used as safe simplification. This model of material is known as: "material perfectly plastic".

Finite element analysis allows nonlinear properties of the material to be quite complex. They even include the true behavior of the steel through the actual calculation of the stress-strain relationship (True Stress - True Strain curve). Nevertheless, when a material behavior is simplified to a perfectly plastic model, it reduces considerably the complexity of the analysis while allowing the redistribution of stresses, a typical property of steel structures. This simplification is practical due to the fact that having the information of the actual strains requires a laboratory test (coupon tensile test). This is why the program uses the nominal efforts (already stipulated in the code). It should also be said that the information required to build the “true stress-true strain curve” is specific from each material used at that time. Furthermore, it may even be different from materials obtained at the same time from same supplier but from different batches.

Figure A - 3: Steel Material Models shows the typical curves used for modeling steel. The curve to be used depends on the type of analysis to be performed. In a linear analysis where the distribution of moments and reactions of slender elements is the information required, the chosen curve should be linearly elastic. If the distribution of stresses in plate-like elements is required, any curve with plastic behavior can be used. In this case, the choice will depend on the accuracy level required.

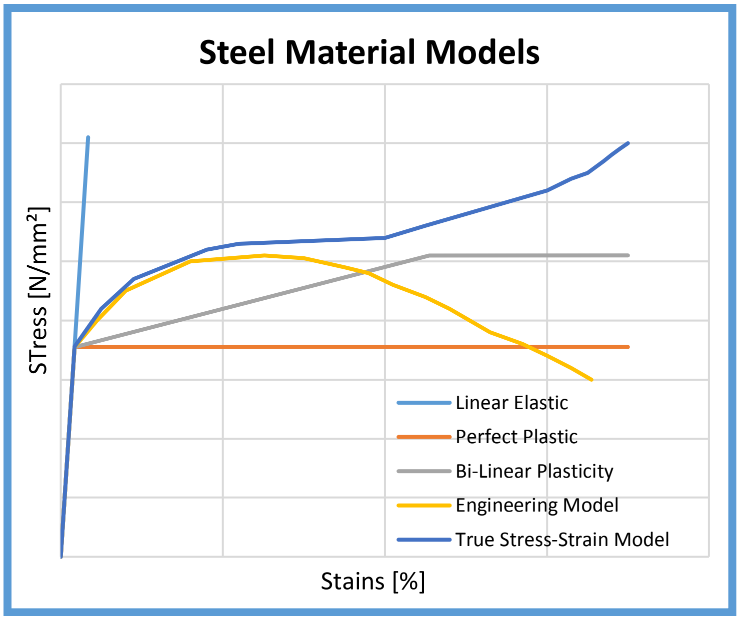


Figure A - 3: Steel Material Models

A.2.4) Finite Element Description

In IDEA StatiCa, only plate elements are available. Moreover, the software simplifies any cross section as a group of plates joined in a specific way. Most of the standard steel profiles are already specified in the software; however, the software also allows the construction of custom sections. Constrains between two plates that form the same cross-section, are automatically performed by the program. The benefit of this characteristic is that the user does not need to worry or spend time in this part, which is a normal stage in a general-purpose finite element software. However, the disadvantage is that the user loses control of the model itself.

Plate elements are important when the distribution of stresses, which follow from the development of non-linear strains in the steel sections, is the request output of the user. This kind of element, in contrast with 1D elements like beams, allows the nonlinear distribution through its height. The consequence of the plate elements is that the program uses 2D finite elements and not 3D volumetric elements as "bricks", which have shown better results to predict the real behavior of bolts (Kim, Yoon, & Kang, 2006) and bearing of plates. An example is the way to model a steel “I” shape section. IDEA StatiCa models the beam as an element formed by three plates where the nodes of the flanges and the web match to be able to generate constrains that allow the proper behavior of the section. A finite element model performed in IDEA StatiCa is shown in Figure A - 4: Finite Element Model of an "I" shape beam.

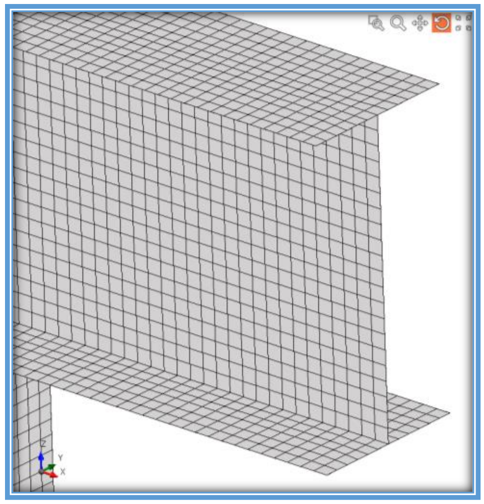


Figure A - 4: Finite Element Model of an "I" shape beam

To generate these plate-like elements, IDEA StatiCa uses shell finite elements. Specifically, it uses a 4-node quadrilateral element called MITC4. The MITC4 is a not flat finite element with its nodes in the corners, which have 6 degrees of freedom [3 translations (ux, uy, uz) & 3 rotations (ϕx, ϕy, ϕz)]. The element was proposed by Edward N. Dvorkin and Klaus-Jürgen Bathe (Dvorkin & Bathe,1984). The formulation starts from three-dimensional continuum description and it is degenerated to shell behavior. It does not depend on a specific shell theory, so it can be used for both, thin and thick shells. The results are very close to those obtained by finite elements derived from shell behavior such as, the Sanders-Koiter equations for thin shells. MITC4 overcomes the "shear locking" problem through a reduced integration formulation that is based on the Mixed Interpolation of Tensional Components (MITC) approach.

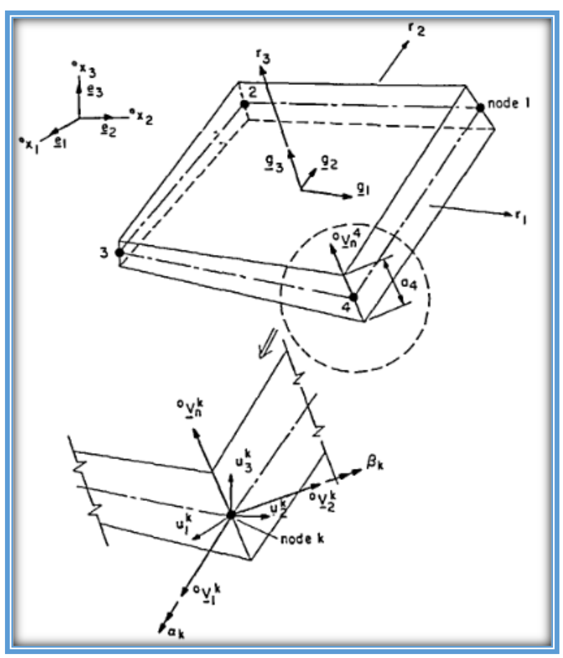


Figure A - 5: MITC4 Element

An advantage of MITC4 is that it is a cost-effective element. This means that the results are accurate enough using small computational capacity. The results are comparable to those elements with 9 and 16 nodes, but which are very expensive in computational terms (Dvorkin & Bathe, 1984). In the different studies, it has been verified that this element also does not suffer from zero energy modes or also known as Hourglass mode. Even though the MITC4 has a very efficient behavior, it is an element sensitive to the size of the mesh. A dense mesh is required in order to obtain results comparable to those of quadratic shell elements (8 or more nodes). (See A.3.4).

During the finite element analysis (FEA), IDEA StatiCa takes into consideration the rotations perpendicular to the plane of the shell finite element. Moreover, the out of plane shear deformation is considered in the formulation of the flexural behavior of the elements based on Mindlin hypothesis.

A.2.5) Welds and Bolts

In the finite element model generated in IDEA StatiCa, the plates of different elements are connected through fasteners, either bolts or welding. In other words, without these connectors the analysis cannot be executed. This is because when the load is applied in one element and the boundary conditions in other element, the transmission of the load between elements is done through the fasteners, without them, this transfer is not possible and the analysis cannot converge. This causes an error and the analysis cannot run.

Now, the way to model these fasteners can be a complex work in any finite element program. In FEA programs for general applications, they can be modeled as volumetric elements. However, in most of the cases, several simplifications are made because the geometry can become very complex. For example, the bolt thread or the real shape of the weld have geometric properties difficult to model; therefore they are replaced by standard geometries like bolts without thread or bolts circular head and nut.

In the case of IDEA StatiCa, the bolts are modeled as massless points that act as a link between the holes of two or more plates. The forces generated at these points are calculated later on an analytical check that is made based on the rules of the code to define the utilization ratio of the bolt.

The analysis process is different for welds. There are two options of analysis for this kind of fasteners in the software. In the first approach, the edge of the plate of one element is connected to one of the faces of the other element. The link generates constrains or ties between the nodes. These constrains are multi-node, where a node belonging to the plate of an element is connected with two or more nodes of the plate of the element to be connected. Subsequently, the stresses are calculated in these constrains and through interpolation, the applied stresses for the weld are obtained. When evaluating these stresses, the user can choose between using the average stress or the maximum stress (see A.3.1) and the selection is compared to the rules of the code.

On the other hand, a plastic welding approach can be executed. In this case, the weld is modeled as a special elasto-plastic element on the multi-node constrain, where the stresses are calculated. Since the weld is modeled as a "volumetric" element, it does not require interpolations to obtain the acting stresses. The calculated stresses are directly checked with the rules stipulated in the code.

A.2.6) Contact

A very important parameter in the finite element analysis of steel joints is the contact between surfaces of the elements. This property is key since the contact can completely change the behavior of a joint. For instance, in a simple shear joint, the geometry and the load can be such, that the rotation of the connected element is too big. This will produce a contact between the plates, generating high compressive stresses. This situation will transform the simple joint into a moment resistant joint.

Modeling the contact constrain is a complex task. It must be only activated when an element touches another, allowing the transfer of compression forces. In the case that two elements are in contact but there is tension, the contact should not generate any link or constrain since the separation of the two elements should be possible. A normal node link does not work in this case. IDEA StatiCa uses the method called penalty stiffness. An advantage of this method is that it is easy to produce convergence in the analysis and the computational cost is low.

After conducting a study in IDEA StatiCa, testing the different types of connections and the different types of contacts between elements, it was observed that this contact is activated only in the presence of fasteners. If the plates of the elements are not connected through fasteners and a compression load is applied, the compression force will not be transferred by simple contact. An example is a column and base plate. If the column is in contact with the base plate, but is not welded to it, and a compression force is applied, it cannot be transferred from the column to the base plate. Therefore, the analysis is not executed.

A.3 Code Parameters

Like any other program of structural design, IDEA StatiCa requires an initial setup in order to be able to function and to deliver the user the desired results. In general, the program is very user-friendly. Learning how to model a joint, how to apply boundary conditions or loads can be done in a very short time. Being a program dedicated exclusively to the design of steel joints, it has very simple parametric modeling tools. By inserting certain data such as length, profile type, material, etc.; the user can model the geometry of a joint in a few minutes, including more complex and non-standard ones. Moreover, if the BIM tools of the program are used, the user can import joints already generated in a CAD program and make the necessary modifications in IDEA.

After the geometry is specified in IDEA StatiCa, the detailing of different types of machining are done in the same way that the real joint will be assembled. The user can choose between cuts, holes, welding, bolts, end plates, fin plates, etc. The whole process is very simple and logical.

However, there is a set of parameters that are general for all the joints and must be specified before the analysis. This parameter group is located in the "Code setup" icon (see Figure A - 6: Code Setup Location). These data regulate the design rules, the desired method of welding analysis, mesh density, among other things. In order to be able to choose the parameters that generate results that are accurate, safe and at a reasonable computational cost, parametric studies were performed. As a result, a set of parameters is proposed. This set is considered to be the most appropriate in terms of accuracy and computational costs. They were tested in the joints studies at chapter 3 and 5 of the present dissertation. Nevertheless, all these parameters are responsibility and taste of the designer.

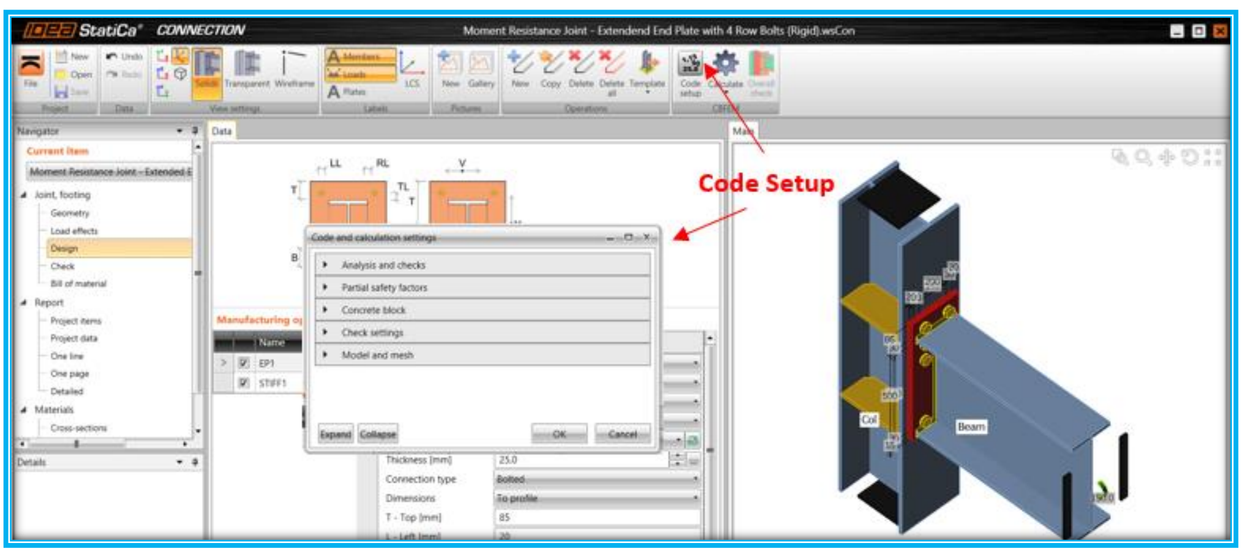


Figure A - 6: Code Setup Location

A.3.1) Analysis and Checks

In this window, the parameters to be chosen are those that will define the method of analysis of the elements such as welding and pre-stressed bolts. The two most interesting and important parameters in this section are described below:

Stop at limit strain: with this option, the analysis can be stopped when a finite element reaches the limit plastic strain, which was previously established. Activating this tool can be very useful if one wants to know the resistance of a connection whose capacity is unknown. This due to the fact that any load can be applied and in case of exceeding the capacity of the connection, the analysis will stop allowing the reduction of computational time in a trial-error process. Another benefit of having this option enabled is the easy identification of the element that is going to fail first, so we can know which one is the weakest component of the joint.

A good way to work is to have this option activated at first. Once the user has an estimate of the joint capacity and the failing component is the desired one, the option must be deactivated and directly enter a load equal to the estimated resistance of the joint. The reason behind this is that if this option is enabled, the analysis will stop when a finite element, which may become very small, reaches the maximum limit of plastic strain. However, this does not mean that the joint or the component has already reached its maximum capacity. Having local yielding does not mean failure and these stresses can be redistributed along the element. If the analysis is allowed to continue, not only this re-distribution is going to happen, but it will also allow the developing of a failure mechanism patron, which will give the actual capacity of the joint.

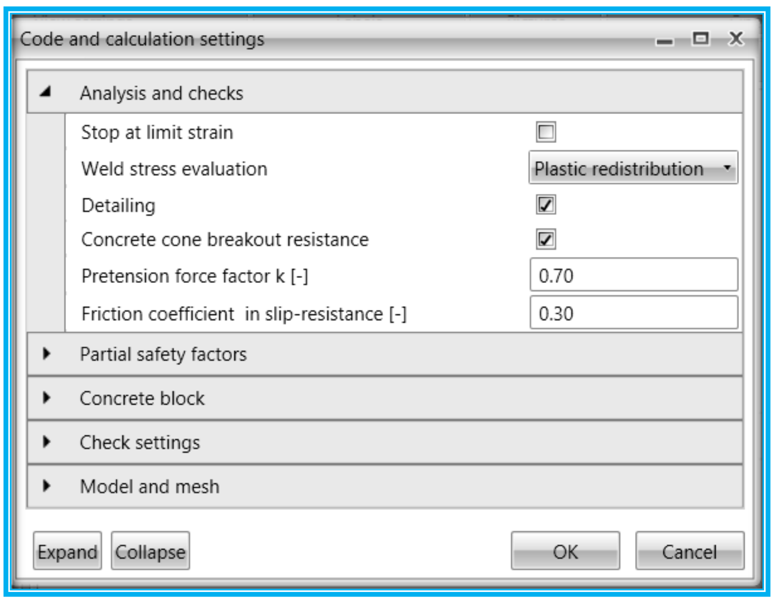


Figure A - 7: Analysis and Check parameters

Weld Stress Evaluation: In point A.2.5, it was mentioned that there are several ways to evaluate the welds. It can be analyzed using interpolation at the multi-constrain points or it can be analyzed by modeling the weld as a virtual volumetric element. In this window the different type of analysis are available to be chosen. The user can pick between using the maximum calculated stress, the calculated average stress or a plastic redistribution of stresses. To understand the difference between maximum value and average stresses, Figure A - 8: Weld Stresses (IDEA StatiCa, 2017) illustrates these values. σ_{FEM} is the stress calculated by FEA.

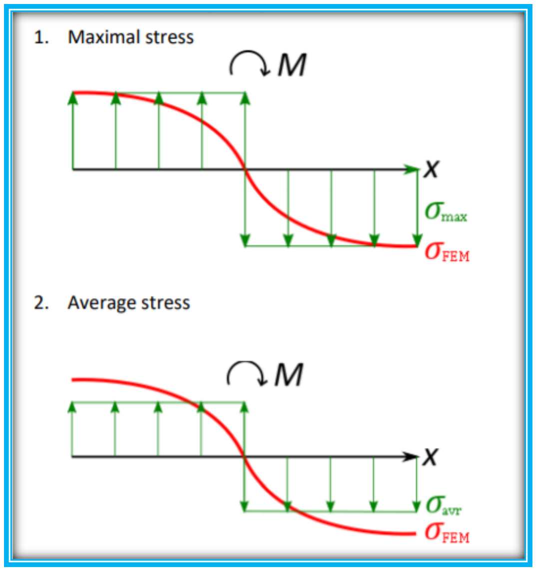


Figure A - 8: Weld Stresses

A.3.2) Partial Safety Factors

Partial safety factors are specified in the code and in the national annexes. These factors give a degree of safety to the structure by affecting the calculated resistance of the elements. In the case of IDEA StatiCa, a finite element analysis is used to calculate the stresses and strains in the components. Partial safety factors do not affect the plate design. The partial safety factors are used to determine the unity check of bolts and welds (and concrete elements). For plates, the design method is based on plastic strains. (See A.3.3).

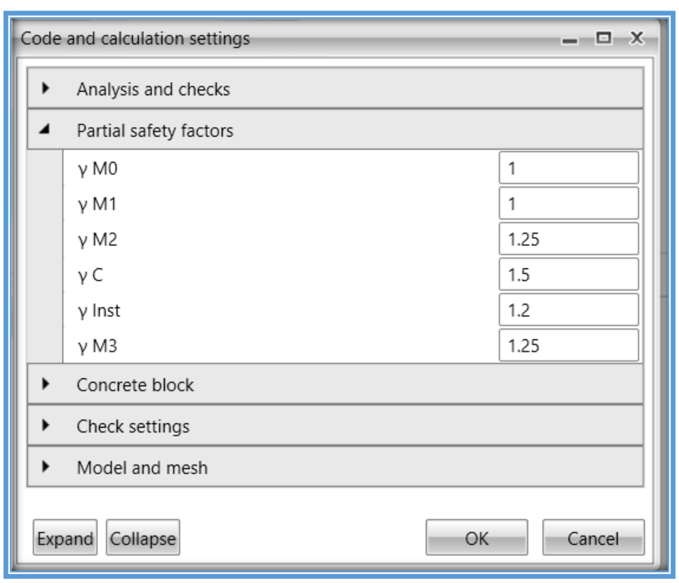


Figure A - 9: Partial Safety Factors Parameters

A.3.3) Check Settings

In this section, the user must enter the design parameters. It is required to stipulate simple parameters. For example, the minimum distance between the bolt hole to the edge of the plate or if the connection is in a sway or no sway frame, among others. The most interesting parameters in this part are:

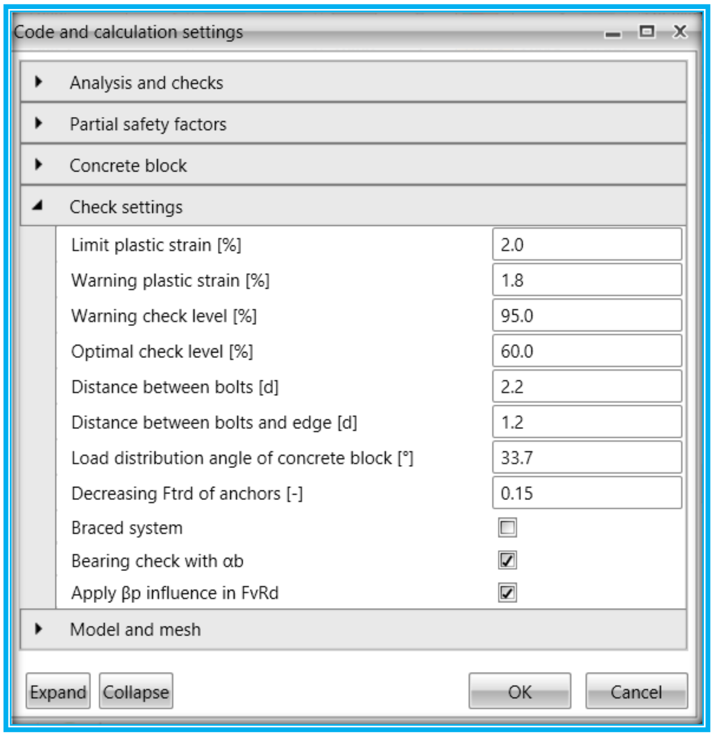


Figure A - 10: Check Settings Parameters

<u>% Check Levels:</u> The check level percentage enables the use of a tool from IDEA StatiCa where an overall check of the plates and fasteners is performed, and the results are given by a color code similar to a traffic light.

a) Green: indicates that the plate or component is within the limits of optimum design.
b) Orange: the use of the plate or component is close to the maximum 100%.
c) Red: The plate or component exceeds the maximum allow percentage of use. Red indicates design error.
d) Gray: the plate or component has a lower utilization rate than the optimum. The design is not efficient.

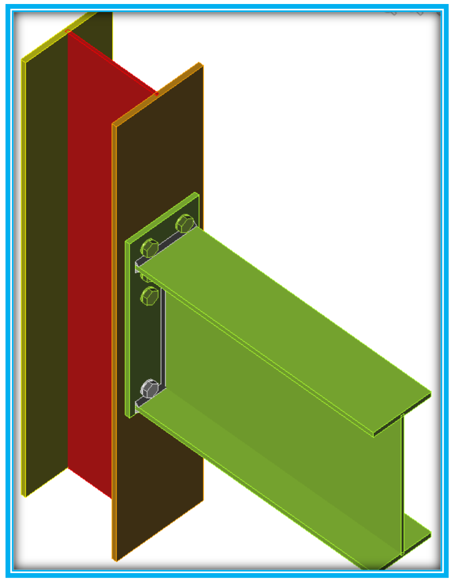


Figure A - 11: Example of the Traffic Light Tool

In figure A - 11, an example of the "Traffic Light Tool" is presented. The joint and its components can be easily evaluated by following the color code. The web of the column is overused, the bottom bolts and the welding are underused, and the rest of the components are within the optimal range of use. It is important to mention that the percentages (optimum, maximum and warning) are fixed earlier in this set of parameters.

Limit Plastic Strain: This is one of the most important and controversial parameters. It defines when can be considered that an element or the entire joint has reached its maximum capacity when FEA is used. A 3% limit of plastic strain in IDEA StatiCa (using the proposed mesh density, see A.3.4) is the value recommended in this thesis (see chapter 5). This 3% value is a local limit. This means that the limit plastic strain can be reached by one or few finite elements without meaning that a global failure has been reached in the joint. When the user is applying the plastic limit, a stiffness analysis will facilitate the extracting of the joint capacity (see A.4.3). In most of the cases, this value does not allow a failure mechanism to be developed. However, the results obtained with this "method" are similar and comparable with the results obtained by following the rules of the Eurocode. The calculated capacity will be at the same safety level as the code.

The developers of the program suggest that in any case, a 5% value for the limit plastic strain should be used. This is due to the fact that the resistance varies very little (4%) when the plastic strain increases from 2% to 5% (František, 2016). The value of 5% is also recommended in EN 1993 1-5, Annex C. However, after performing the analysis of several joints, it was determined that this is not accurate for all types of joints. There are cases where the variation of resistance can be bigger. As it is the case of the joint shown in figure A-12. The curve was obtained from a semi-rigid bolted joint with extended end plate. It can be seen that the resistance varies by 27% when changing from 2% to 5% of plastic strain. With this evidence, the designer must evaluate the value for the limit of the plastic strain allowed in the elements.

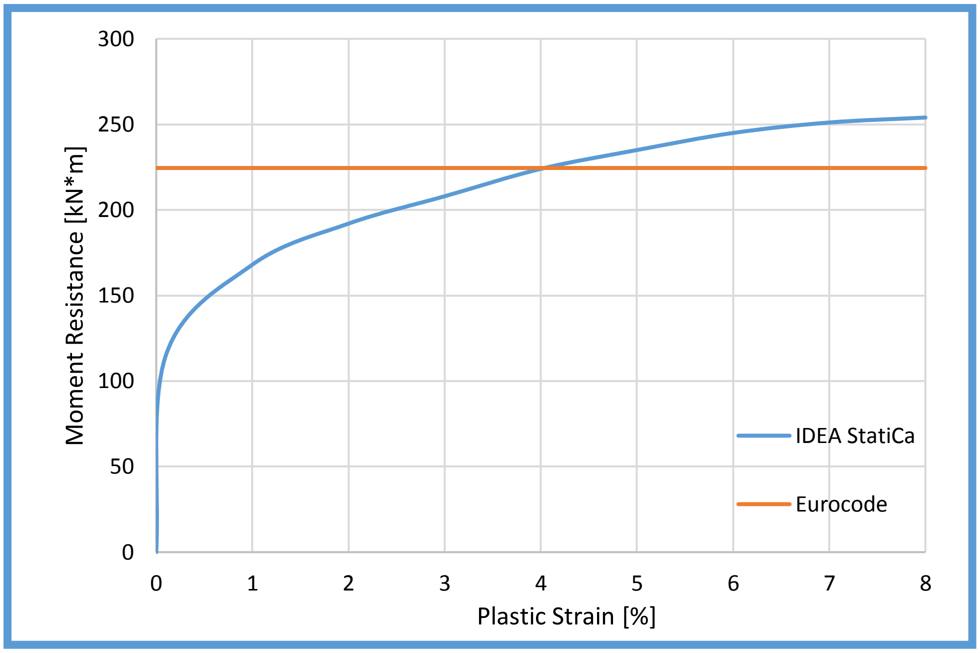


Figure A - 12: Moment vs Plastic Strain Curve

A.3.4) Model and Mesh

In this window, the density of the mesh is controlled. IDEA StatiCa does not allow the user to modify the shape, location, and type of element in the finite mesh. Neither has advanced mesh properties like a tool to apply high mesh density in strategic locations of the model and low mesh density where the analysis is not complex or the results are not important. The user has no control over the nodes where the constrains are to be applied either by contact or by applying a fastener. These properties are generated automatically. The only parameter that can be controlled is the number of elements. Here is where the mesh density can be increased and therefore it allows the user to have some control to get accurate results. It is well known that at higher density, the accuracy is better, but also the computational costs are higher.

It is precisely this characteristic that shows that IDEA StatiCa is reliable for daily practice in the design of steel joints, but it is not recommended for academic-experimental purposes. The program does not provide the necessary information for analysis and does not allow the correct manipulation of the finite element mesh, which means that the necessary information is not available to determine if the failure mechanism is adequate. It is hard to conclude if the yielding is local or global. This characteristic must be evaluated by the designer.

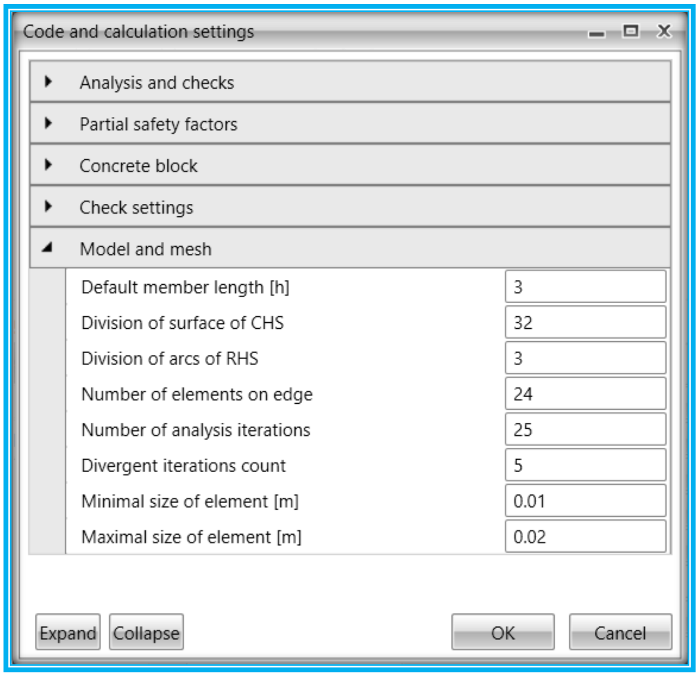


Figure A - 13: Model Mesh Parameters

__Number of Elements on Edge:__ This parameter controls the number of elements to be generated along the critical edge. The program performs the selection of the critical edge automatically. When increasing the number of elements it is important that the maximum and minimum size of the finite element is adequate (congruent) with the number of elements. In this way the desired number of elements can be obtained.

The program developers recommend using eight elements in the critical edge. After a parametric study, it was concluded that the number due to computational cost, efficiency and accuracy, is 24 elements at the critical edge, using a maximum element size of 2cm and a minimum of 1cm. Figure A-14 shows the parametric study for a semi-rigid joint with flush end plate. It can be observed that an element size smaller than 20mm has a considerably effect in the analysis time. On the other hand, the resistance only varies in the order of 10%.

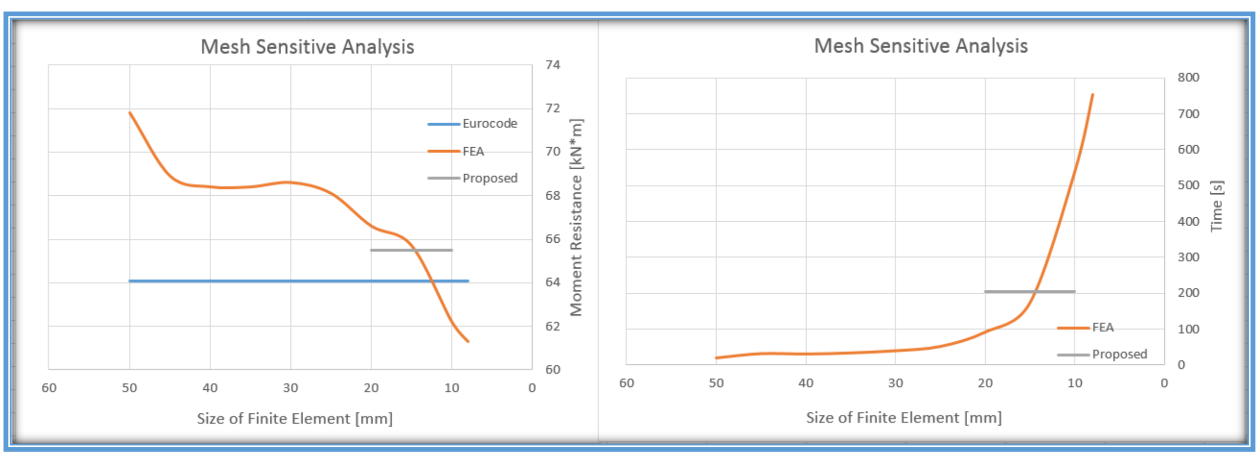


Figure A - 14: Mesh Sensitive Analysis

The computational time increase considerably when the mesh size is reduce. Nevertheless, for a simple standard joint, even with a fine mesh, the computational time does not exceed half an hour. If we compare this time with ABAQUS, the same joint with the same parameters will need approximately 3 hours of analysis. This fast analysis is one of the benefits that the developers promote about using IDEA StatiCa. The low computational cost makes this finite element tool suitable for engineer daily practice.

The mesh size in IDEA StatiCa is controlled principally by the size limitations. The user need to specify the maximum and the minimum finite element edge size. In order to set the correct boundaries and check that the selected mesh is the correct a simple way to check will be through a parametric study were the maximum and the minimum limits are define and the variation of the element number is the changing parameter.

The proposed setup of 24 elements in the critical edge with mesh size between 20 to 10mm, and a limit plastic strain of 3%, has a good approximation to the Eurocode solution and the computational time is reasonable. It can be seen in Figure A - 14: Mesh Sensitive Analysis that the proposed setup time is just above the point where the time variation became exponential.

A.4 Type of Analysis

Once the parameters for a suitable setup have been chosen, the type of analysis to perform need to be selected. Each type of analysis delivers specific information. Depending on what kind of verification is required, the correct choice of type of analysis can save time and facilitate the efficient use of the program. The type of analysis also influences the computational time. For example, the Stress/Strain Analysis (EPS) takes approximately one fifth of the time for a Stiffness (ST) analysis. The types of analysis and their implications are detailed below:

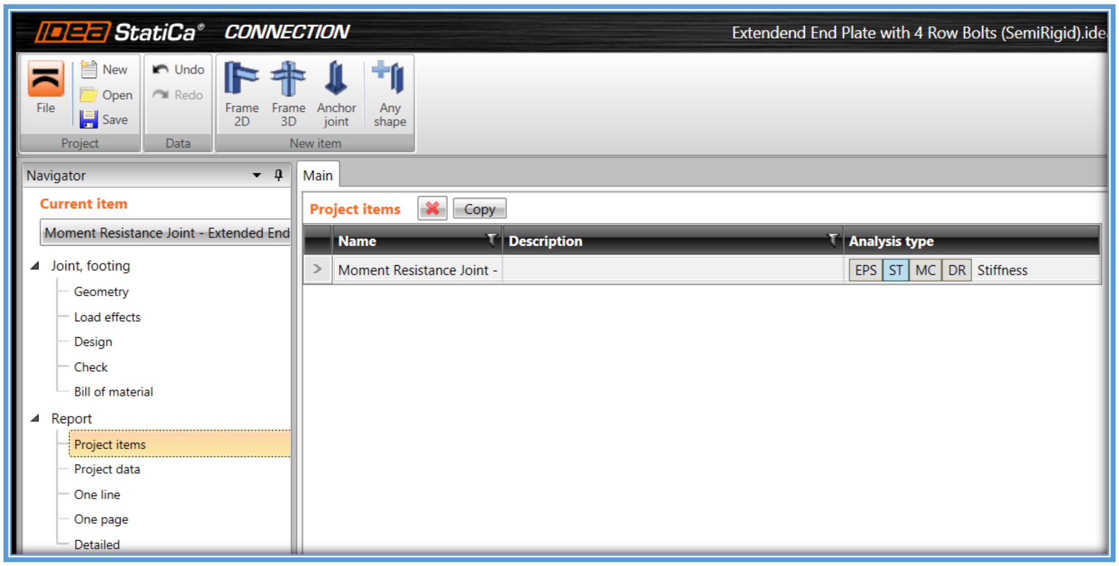


Figure A - 15: Types of Analysis Selection Window

A.4.1) Stress/Strain Analysis (EPS)

With this analysis, a physically non-linear study is performed. This means that the properties of the material are non-linear (perfectly plastic). The load is applied gradually and the iterations are executed to find the convergence / equilibrium of stresses and strains. This analysis has low computational cost and allows us to understand and visualize the distribution of stresses and strains. A correct interpretation of the results can help the designer to determine the failure mechanism that is governing the element/joint design (see chapter 5).

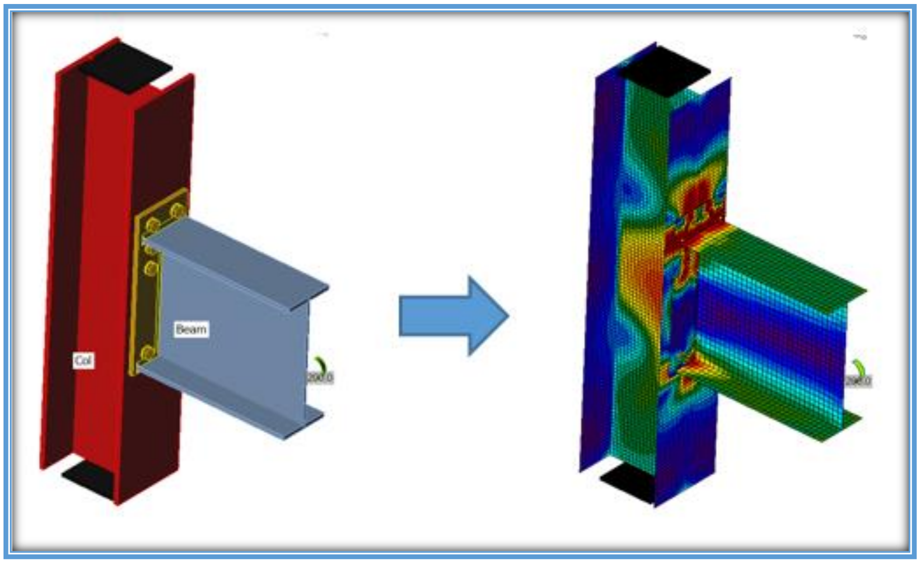


Figure A - 16: Stress/Strain Analysis

A.4.2) Stability (Buckling) Analysis (EPS)

IDEA StatiCa can perform a linear buckling analysis. Graphically, it is possible to observe the place where the buckling is going to occur, which allows analyzing if it is a local or global buckling of the joint. The program delivers the critical loading factors and the buckling shapes. The number of eigenmodes cannot be specified by the user. In terms of computational cost, this analysis is efficient and fast. However, it is mesh-sensitive so a parametric study to measure this sensitivity is always recommended. The recommended value of 24 elements (maximal element size of 2cm and minimum of 1cm) at the critical edge, proved to deliver acceptable results.

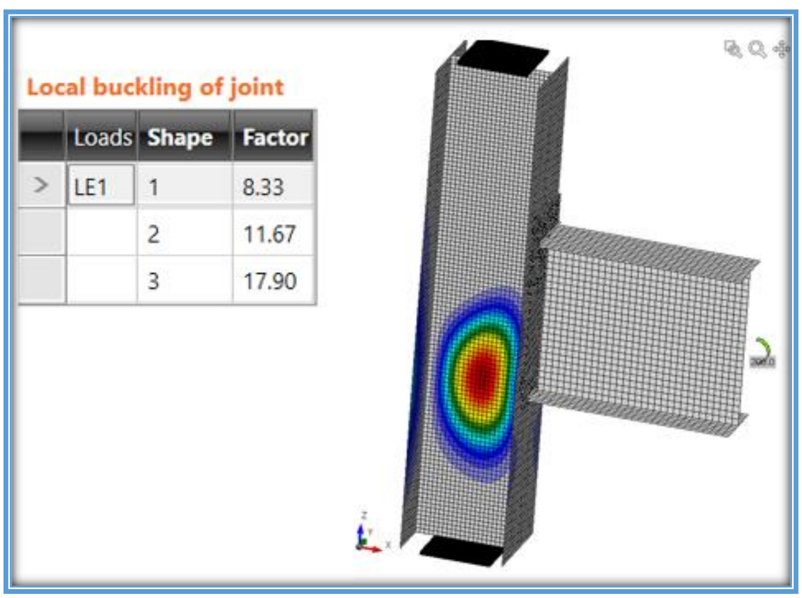
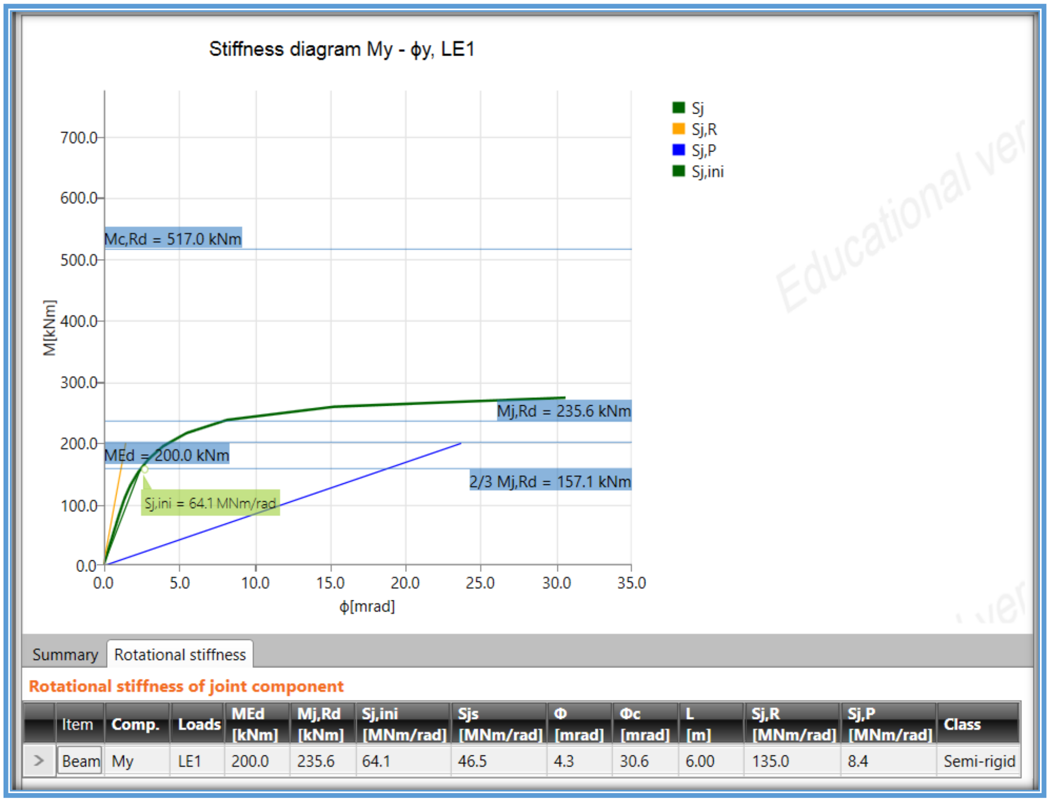


Figure A - 17: Buckling Analysis

A.4.3) Stiffness Analysis (ST)

The program uses FEA to obtain the current rigidity of the different components of the joint. Later, based on the component method given in the Eurocode, it delivers the classification of the steel joint according to its rigidity (pinned, semi-rigid or rigid). For this, several parameters of the code are calculated and their results are tabulated. In addition, the program generates moment vs. rotation curves, where the designer can visualize the behavior of the stiffness and the graphical value of the code limits for joint classification.

Figure A - 18: Stiffness Analysis



Item	Comp.	Loads	MEd [kNm]	Mj,Rd [kNm]	Sj,ini [MNm/rad]	Sjs [MNm/rad]	Φ [mrad]	Φc [mrad]	L [m]	Sj,R [MNm/rad]	Sj,P [MNm/rad]	Class
Beam	My	LE1	200.0	235.6	64.1	46.5	4.3	30.6	6.00	135.0	8.4	Semi-rigid

Figure A - 18: Stiffness Analysis

You can see in Figure A - 18: Stiffness Analysis the graph and the table where the following relevant information is given:

- Limit value of capacity of the joint.
- 2/3 of limit capacity for calculation of initial stiffness
- Value of initial stiffness (Sj, ini)
- Value of secant stiffness (Sjs)
- Limits for the classification of the joint (rigid and pinned)
- Rotational deformation (Φ)
- Rotational capacity (Φc)

When the stiffness analysis is performed, it is applicable individually to an element of the joint. This member is the one that is going to be under analysis. If the joint has several members (e.g. 3D joints), the program fixes the end of all the members (bearing properties), except for the analyzed element, where the load should be applied. In this way, the stiffness of the element under study is not affected by the deformation of the other members, but only by the stiffness of the node and its own rigidity. This is the analysis that takes more computational time. The graphs and results are comparable to those that can be obtained following the calculations of the Eurocode rules. An important disadvantage of this analysis is that the user cannot extract the different points of the curve, so the curve cannot be replicated and the exact value of the individual points cannot be extracted. Graphic data and file are not compatible with other programs, e.g. CAD software.

A.4.4) Member Capacity Design (MC)

The objective of this analysis is to analyze the behavior/capacity of the joint when a M_{max} moment is applied. This "M_{max}" moment is the maximum moment that the analyzing member can transfer to the connection. This moment cannot be modified by the user and is calculated as follows:

$M_{\max} = M_{pl} * 1.1 * \gamma V$ where, M_{pl} is the plastic moment of the analyzed member.

1.1*γV, are factors of moment amplification, to take into consideration uncertainties of various sources of over-strength, stipulated in EN 1998

During this analysis, the joint and its components are checked, excluding the analyzed element. If the designer want to classify the joint as full strength joint, it must resist a moment greater than the maximum moment transferred. This tool is very useful for seismic zones, where it is often required or desired that the joints are able to withstand the maximum moment transferred by a member (usually the beam) so that the plastic hinge can develop in that member. The development of plastic hinges in specific locations are important properties in an earthquake resistant structure.

A.4.5) Design Joint Resistance

During the normal practices in the steel joint design, the designer wants to know the maximum capacity of the steel joint. With the Design Joint Resistance tool, the program gives, in percentage, the ratio between the maximum capacity of the joint with respect the applied load. In this way, the user can appreciate the level of use of the joint. If a percentage less than 100% is obtained, the joint can resist the applied load. While if 100% is exceeded, the connection is failing. This tool is a quick unit check, which delivers as a product a graph indicating the level of plastic strain vs unit check. It also delivers the maximum plastic strain that occurs at the connection.

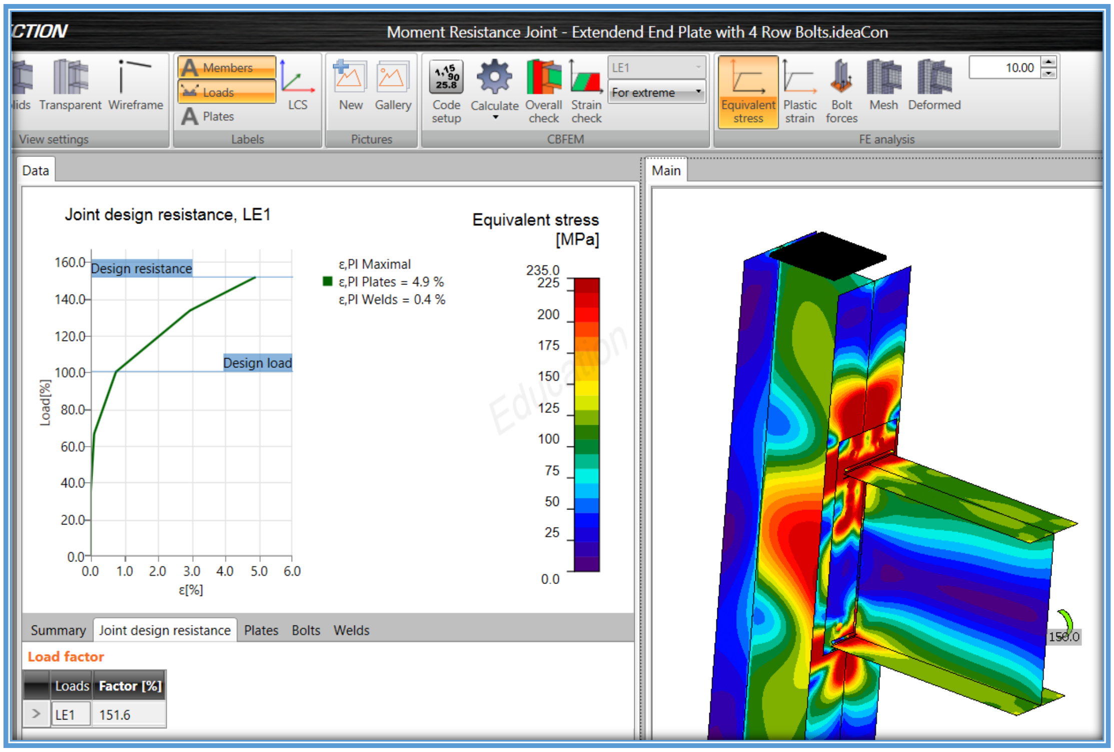


Figure A - 19: Joint Capacity Design

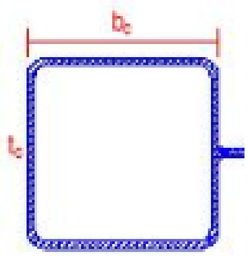
Annex B – Hand Calculation for the Experiment Joint (No Safety Factors)

Input Code Parameters:

$\gamma_{M0} := 1.0 \quad \gamma_{M1} := 1.0 \quad \gamma_{M2} := 1.25 \quad E := 210000 \ \frac{N}{mm^2}$

Section Properties and Geometry:

Column: SHS 220x220x8 (S690)



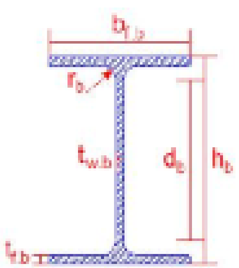
$b_{f.c} := 202\ mm$

$t_c := 8\ mm$ $\quad f_{y.c} := 690\ MPa$

$r_{1.c} := 20\ mm$ $\quad f_{u.c} := 770\ MPa$

$r_{2.c} := 12\ mm$

Beamn: IPE 400 (S355)



$h_b := 402\ mm$ $\quad r_b := 21\ mm$ $\quad I_b := 1318\ cm^4$

$b_{f.b} := 177\ mm$ $\quad d_b := 334.5\ mm$ $\quad A_b := 84.50\ cm^2$

$t_{w.b} := 8.65\ mm$ $\quad f_{y.b} := 355\ MPa$

$t_{f.b} := 12.75\ mm$ $\quad f_{u.b} := 510\ MPa$

Fin Plate:

$h_p := 330\ mm$

$b_p := 200\ mm$

$t_p := 10\ mm$

$f_{y.p} := 355\ MPa$

$f_{u.p} := 510\ MPa$

Bolts: M24 - Grade 10.9

$d := 24\ mm$ $\quad n_1 := 3$

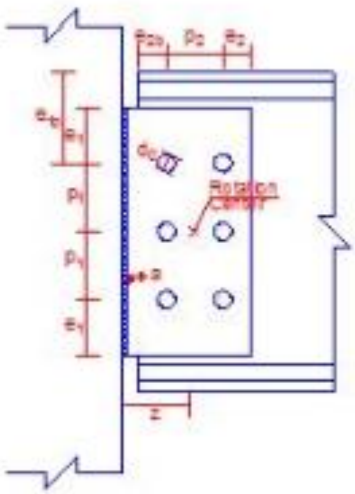
$d_0 := 26\ mm$ $\quad n_2 := 2$

$A_s := 353\ mm^2$ $\quad n := n_1 \cdot n_2 = 6.00$

$f_{y.bo} := 900\ MPa$ $\quad \alpha_v := 0.5$

$f_{u.bo} := 1000\ MPa$

Joint Geometry:



$e_1 := 75\ mm$ $\quad e_2 := 60\ mm$ $\quad z := 102.5\ mm$

$p_1 := 90\ mm$ $\quad e_{2b} := 45\ mm$ $\quad g_h := 20\ mm$

$e_{1b} := 110\ mm$ $\quad p_2 := 75\ mm$ $\quad h_e := \dfrac{h_b - h_p}{2} = 36.00\ mm$

$e_2 + p_2 + e_{2b} + g_h = 200.00\ mm$

Calculations:

1.- Shear Resistance of Bolts ($V_{RD.1}$):

$$F_{v.bo.RD} := \frac{\alpha_v \cdot A_s \cdot f_{u.bo}}{\gamma_{M2}} = 141.20\ kN$$ **Shear Resistance of a Single Bolt ($F_{v.RD}$)**

$$I := \frac{n_1}{2} \cdot p_2^{\ 2} + \frac{n_1 \cdot \left(n_1^{\ 2} - 1\right) \cdot p_1^{\ 2}}{6} = 40837.50\ mm^2$$

$$V_{RD.1} := n \cdot F_{v.bo.RD} = 847.20\ kN$$

2.- Bearing Resistance of the Fin Plate ($V_{RD.2}$):

Bearing Resistance in Vertical Direction:

$$k_{1.p.ver} := min\left(\frac{2.8 \cdot e_2}{d_0} - 1.7\right) = 4.76$$

$$\alpha_{b.p.ver} := min\left(\frac{e_1}{3 \cdot d_0}, \frac{p_1}{3 \cdot d_0} - \frac{1}{4}\right) = 0.90$$

$$F_{b.p.ver} := \frac{k_{1.p.ver} \cdot \alpha_{b.p.ver} \cdot f_{u.p} \cdot d \cdot t_p}{\gamma_{M2}} = 421.42\ kN$$

Bearing Resistance in Horizontal Direction:

$$k_{1.p.hor} := min\left(\frac{2.8 \cdot e_1}{d_0} - 1.7, \frac{1.4 \cdot p_1}{d_0} - 1.7\right) = 3.15$$

$$\alpha_{b.p.hor} := min\left(\frac{e_2}{3 \cdot d_0}\right) = 0.77$$

$$F_{b.p.hor} := \frac{k_{1.p.hor} \cdot \alpha_{b.p.hor} \cdot f_{u.p} \cdot d \cdot t_p}{\gamma_{M2}} = 236.98\ kN$$

Bearing Resistance of the Fin Plate:

$$\beta := \frac{z \cdot (n_1 - 1) \cdot p_1}{I \cdot 2} = 0.23 \qquad \alpha := \frac{z}{I} \cdot \frac{p_2}{2} = 0.09$$

$$V_{RD.2} := \frac{1}{\sqrt{\left(\frac{\frac{1}{n} + \alpha}{F_{b.p.ver}}\right)^2 + \left(\frac{\beta}{F_{b.p.hor}}\right)^2}} = 879.90\ kN$$

3.- Shear Resistance of the Fin Plate: Gross Section (VRD.3):

$$V_{RD.3} := \frac{300\ mm \cdot t_p \cdot f_{y.p}}{\sqrt{3} \cdot \gamma_{M0}} = 614.88\ kN$$

4.- Shear Resistance of the Fin Plate: Net Section (VRD.4):

$$A_{v.p.net} := t_p \cdot (h_p - n_1 \cdot d_0) = 2520\ mm^2$$

$$V_{RD.4} := \frac{A_{v.p.net} \cdot f_{u.p}}{\sqrt{3} \cdot \gamma_{M2}} = 593.61\ kN$$

5.- Shear Resistance of the Fin Plate: Shear Block (VRD.5):

$$A_{nt.p} := t_p \cdot \left(p_2 + e_2 - \frac{3 \cdot d_0}{2}\right) = 960\ mm^2$$

$$A_{nv.p} := t_p \cdot \left(h_p - e_1 - \left(n_1 - 0.5\right) \cdot d_0\right) = 1900\ mm^2$$

$$V_{RD.5} := \frac{f_{u.p} \cdot A_{nt.p}}{\gamma_{M2}} + \frac{f_{y.p} \cdot A_{nv.p}}{\sqrt{3} \cdot \gamma_{M0}} = 781.10\ kN$$

6.- Bending Resistance of the Fin Plate ($V_{RD.6}$):

$$W_{el.p} := \frac{t_p \cdot h_p^{\ 2}}{6} = 181500\ mm^3$$

$$V_{RD.6} := \frac{W_{el.p} \cdot f_{y.p}}{z \cdot \gamma_{M0}} = 628.61\ kN$$

7.- Buckling Resistance of the Fin Plate ($V_{RD.7}$):

$$\lambda_{LT.p} := 2.8 \cdot \sqrt{\frac{z \cdot h_p}{1.5 \cdot t_p^{\ 2}}} = 42.05 \qquad f_{LT.p} := 285\ \frac{N}{mm^2}$$

Information taken from table 17 of BS5950-1.

$$V_{RD.7} := \begin{array}{l} \text{if } z > \dfrac{t_p}{0.15} \\ \quad \left\| \begin{array}{l} \text{if } \dfrac{W_{el.p} \cdot f_{LT.p}}{z \cdot 0.6 \cdot \gamma_{M1}} > \dfrac{W_{el.p} \cdot f_{y.p}}{z \cdot \gamma_{M0}} \\ \quad \left\| V_{RD.6} \right. \\ \text{else} \\ \quad \left\| \dfrac{W_{el.p} \cdot f_{LT.p}}{z \cdot 0.6 \cdot \gamma_{M1}} \right. \end{array} \right. \\ \text{else} \\ \quad \left\| V_{RD.6} \right. \end{array} = 628.61\ kN$$

8.- Bearing Resistance of the Beam Web ($V_{RD.8}$):

Bearing Resistance in Vertical Direction:

$$k_{1.b.ver} := min\left(\frac{2.8 \cdot e_{2b}}{d_0} - 1.7\right) = 3.15$$

$$\alpha_{b.b.ver} := min\left(\frac{p_1}{3 \cdot d_0} - \frac{1}{4}\right) = 0.90$$

11.- Shear Resistance of Beam Web: Shear Block ($V_{RD.11}$):

$$A_{nt.b} := t_{w.b} \cdot \left(e_{2b} - \frac{d_0}{2}\right) = 277\ mm^2$$

$$A_{nv.b} := t_{w.b} \cdot \left(e_{1b} + (n_1 - 1) \cdot p_1 - (n_1 - 0.5) \cdot d_0\right) = 1946\ mm^2$$

$$V_{RD.11} := \frac{f_{u.b} \cdot A_{nt.b}}{\gamma_{M2}} + \frac{f_{y.b} \cdot A_{nv.b}}{\sqrt{3} \cdot \gamma_{M0}} = 511.84\ kN$$

12.- Shear Resistance of Columns Face ($V_{RD.12}$):

$$\text{if } \frac{(b_{f.c} - 4 \cdot t_c)}{t_c} < 1.4 \cdot \sqrt{\frac{E}{f_{y.c}}} = \text{"OK"}$$

"OK"
else
"Slender Column Face"

$$V_{RD.12} := \frac{2 \cdot h_p \cdot t_c \cdot \sqrt{3} \cdot f_{y.c}}{\gamma_{M2}} = 5048.17\ kN$$

Expresion given by:
DESIGN GUIDE 9 - STRUCTURAL HOLLOW SECTION COLUMN CONNECTIONS

13.- Shear Resistance of the Joint (V_{RD}):

$$V_{RD} := \min\left(V_{RD.1}, V_{RD.2}, V_{RD.3}, V_{RD.4}, V_{RD.5}, V_{RD.6}, V_{RD.7}, V_{RD.8}, V_{RD.9}, V_{RD.10}, V_{RD.11}, V_{RD.12}\right)$$

$$\boldsymbol{V_{RD} = 511.84\ kN}$$

Welds Calculation:

Welds S355:

$V_{ED.w} := 365.5\ kN$

$M_{ED.w1} := V_{ED.w} \cdot z = 37.46\ kN \cdot m$ $\qquad$ $M_{ED.w2} := 15\ kN \cdot m$

$M_{ED.w} := \min\left(M_{ED.w1}, M_{ED.w2}\right)$

$l_w := 316\ mm$ $\qquad$ $\beta_w := 0.90$ $\qquad$ $a := 3\ mm$ $\qquad$ $f_{u.weld} := 580\ \frac{N}{mm^2}$

$$\tau_2 := \frac{V_{ED.w}}{2 \cdot a \cdot l_w} = 192.77 \ \frac{N}{mm^2}$$

$$\tau_1 := \frac{3 \cdot M_{ED.w}}{\sqrt{2} \cdot a \cdot {l_w}^2} = 106.22 \ \frac{N}{mm^2}$$

Expresion for Two Moment Method. See below figure.

$$\sigma_1 := \tau_1 = 106.22 \ \frac{N}{mm^2}$$

$$\sigma_w := \sqrt{{\sigma_1}^2 + 3 \cdot (\tau_1 + \tau_2)^2} = 528.65 \ \frac{N}{mm^2}$$

$$f_{RD.w} := \frac{0.75 \cdot f_{u.weld} + 0.125 \cdot f_{u.c} + 0.125 \cdot f_{u.b}}{\beta_w \cdot \gamma_{M2}} = 528.89 \ \frac{N}{mm^2}$$

$$\left.\begin{array}{l} \text{if } f_{RD.w} < \sigma_w \\ \quad \| \text{"Weld Failure"} \\ \text{else} \\ \quad \| \text{"OK"} \end{array}\right| = \text{"OK"}$$

Annex C – Hand Calculations for Semi-Rigid and Rigid Joints Design Resistance of Chapter 5

C.1 Semi-Rigid Joint

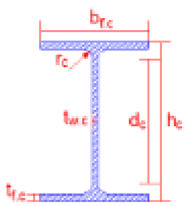
Imput Parameters:

$\gamma_{M0} := 1.0$ $\gamma_{M1} := 1.1$ $\gamma_{M2} := 1.25$ $\beta := 1.0$

$l_b := 1300\ m$ $E := 210000\ \dfrac{N}{mm^2}$

β is the trasnformation parameter (Tab 5.4).

Section Properties and Geometry:
Column (HEM 450):



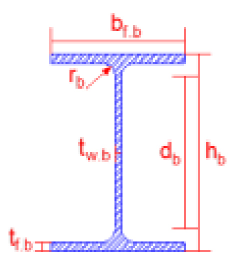
$h_c := 478\ mm$ $r_c := 27\ mm$ $A_c := 33540\ mm^2$

$b_{f.c} := 307\ mm$ $d_c := 344\ mm$

$t_{w.c} := 21\ mm$ $f_{y.c} := 275\ MPa$ $\varepsilon_c := \sqrt{\dfrac{235\ MPa}{f_{y.c}}} = 0.92$

$t_{f.c} := 40\ mm$ $f_{u.c} := 360\ MPa$

Beamn (IPE 400):



$h_b := 400\ mm$ $r_b := 21\ mm$ $I_b := 23.13 \cdot 10^7\ mm^4$

$b_{f.b} := 180\ mm$ $d_b := 331\ mm$ $A_b := 8450\ mm^2$

$t_{w.b} := 8.6\ mm$ $f_{y.b} := 275\ MPa$ $\varepsilon_b := \sqrt{\dfrac{235\ MPa}{f_{y.b}}} = 0.92$

$t_{f.b} := 13.5\ mm$ $f_{u.b} := 360\ MPa$

$W_{pl.b} := 1307000\ mm^3$

End Plate:

$h_p := 386\ mm$

$b_p := 180\ mm$

$t_p := 10\ mm$

$f_{y.p} := 294\ MPa$

$f_{u.p} := 360\ MPa$

Bolts: M24 - Grade 10.9

$d := 24\ mm$ $n_1 := 2$ $t_{wa} := 2\ mm$

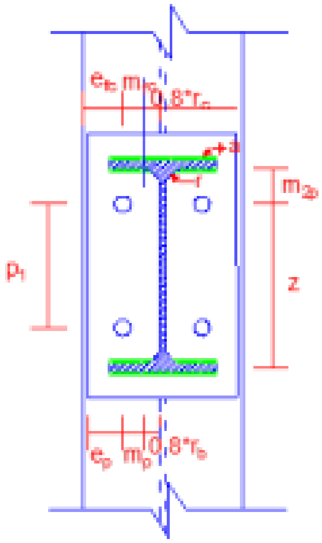
$d_0 := 26\ mm$ $n_2 := 2$

$A_s := 353\ mm^2$ $n_{t.bo} := 4$

$f_{y.bo} := 900\ MPa$ $t_{h.bo} := 0.7 \cdot d = 16.8\ mm$

$f_{u.bo} := 1000\ MPa$ $t_{w.bo} := 0.8 \cdot d = 19.2\ mm$

Joint Geometry:



$e_{fc} := 103.5\ mm$ $\quad e_p := 40\ mm$ $\quad p_1 := 200\ mm$

$z := 293.25\ mm$ $\quad z_2 := z - p_1$ $\quad e_{min.fc} := e_p$

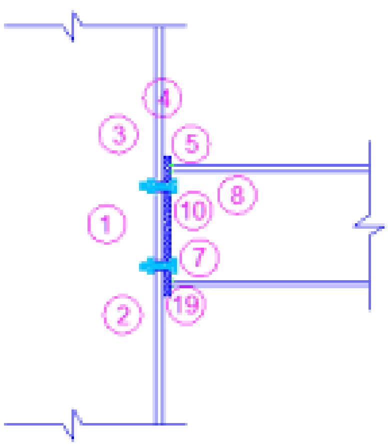
$a_f := 6\ mm$ $\quad a_w := 4\ mm$ $\quad \beta_w := 0.9$

$n_p := e_p$ $\quad m_p := 45.7\ mm - \sqrt{2} \cdot a_w = ?\ mm$

$n_{fc} := e_{min.fc}$ $\quad m_{2.p} := 86.5\ mm - \sqrt{2} \cdot a_f = 78.01\ mm$

$$m_{fc} := \frac{b_{f.c}}{2} - e_{fc} - \frac{t_{w.c}}{2} - 0.8 \cdot r_c = 17.90\ mm$$

Calculation of the Individual Componets:



1 - Column Web Panel in Shear
2 - Column Web in Transverse Compression
3 - Column Web in Tension
4 - Column Flange in Bending
5 - End Plate in Bending
7 - Beam Web/Flange in Compresion
8 - Beam Web in Tension
10 - Bolts in Tension
19 - Welds

1.- Column Web Panel in Shear (1):

a) Resistance:

$$A_{v.c} := A_c - 2 \cdot b_{f.c} \cdot t_{f.c} + \left(t_{w.c} + 2 \cdot r_c\right) \cdot t_{f.c} = 11980\ mm^2$$

$$V_{wp.RD} := \frac{0.9 \cdot f_{y.c} \cdot A_{v.c}}{\sqrt{3} \cdot \gamma_{M0}} = 1711.87 \; kN$$

Limitation:

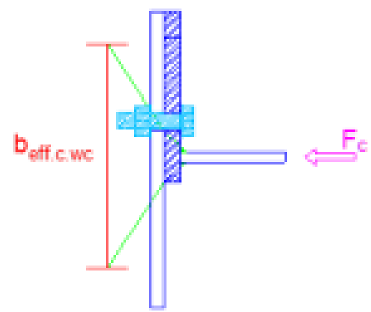
$$\text{if } \frac{d_c}{t_{w.c}} \le 69 \cdot \sqrt{\frac{235 \; MPa}{f_{y.c}}} \;\; \| \; \text{“OK”} \;\; \text{else} \;\; \| \; \text{“Web Slender too High”} \quad = \text{“OK”}$$

$$F_{1.RD} := \frac{V_{wp.RD}}{\beta} = 1711.87 \; kN$$

b) Stiffness:

$$k_1 := \frac{0.38 \cdot A_{v.c}}{\beta \cdot z} = 15.52 \; mm$$

2.- Column Web in Transverse Compression (2):



a) Resistance Stregth:

$$s := r_c \qquad s_p := 2 \cdot t_p = 20.00 \; mm$$

$$b_{eff.c.wc} := t_{f.b} + 2 \cdot \sqrt{2} \cdot a_f + 5 \cdot (t_{f.c} + s) + s_p = 385.5 \; mm$$

$$\omega := \frac{1}{\sqrt{1 + 1.3 \left(\frac{b_{eff.c.wc} \cdot t_{w.c}}{A_{v.c}} \right)^2}} = 0.79 \qquad k_{w.c} := 1.0$$

$$\lambda_p := 0.932 \cdot \sqrt{\frac{b_{eff.c.wc} \cdot d_c \cdot f_{y.c}}{E \cdot t_{w.c}^{\;2}}} = 0.58$$

Compress

$$\rho := \text{if } \lambda_p > 0.72 \;\; \left\| \; \frac{(\lambda_p - 0.2)}{\lambda_p^{\;2}} \right. \;\; \text{else} \;\; \| \; 1 \quad = 1.00$$

$$F_{2.RD} := min \left(\frac{\omega \cdot k_{w.c} \cdot b_{eff.c.wc} \cdot t_{w.c} \cdot f_{y.c}}{\gamma_{M0}}, \frac{\omega \cdot k_{w.c} \cdot \rho \cdot b_{eff.c.wc} \cdot t_{w.c} \cdot f_{y.c}}{\gamma_{M1}} \right) = 1603.13 \; kN$$

b) Stiffness:

$$k_2 := \frac{0.7 \cdot b_{eff.c.wc} \cdot t_{w.c}}{d_c} = 16.47 \; mm$$

3.- Column Web in Transverse Tension (3):

a) Resistance Stregth:

As Individual Row Resistance:

$$b_{eff.t.I.wc} := min\left(2 \cdot \pi \cdot m_{fc}, 4 \cdot m_{fc} + 1.25 \cdot e_{fc}\right) = 112\ mm$$

$$\omega_1 := \frac{1}{\sqrt{1 + 1.3\left(\frac{b_{eff.t.I.wc} \cdot t_{w.c}}{A_{v.c}}\right)^2}} = 0.98$$

$$F_{3.I.RD} := \frac{\omega_1 \cdot b_{eff.t.I.wc} \cdot t_{w.c} \cdot f_{y.c}}{\gamma_{M0}} = 633.70\ kN$$

As Group 1-2 Resistance:

$$l_{eff.G12.cp.R1} := 2 \cdot p_1 = 400.00\ mm \qquad l_{eff.G12.cp.R2} := 2 \cdot p_1 = 400.00\ mm$$

$$l_{eff.G12.nc.R1} := p_1 = 200.00\ mm \qquad l_{eff.G12.nc.R2} := p_1 = 200.00\ mm$$

$$b_{eff.t.wc} := min\left(l_{eff.G12.cp.R1}, l_{eff.G12.nc.R1}\right) + min\left(l_{eff.G12.cp.R2}, l_{eff.G12.nc.R2}\right) = 400\ mm$$

$$\omega_1 := \frac{1}{\sqrt{1 + 1.3\left(\frac{b_{eff.t.wc} \cdot t_{w.c}}{A_{v.c}}\right)^2}} = 0.78$$

$$F_{3.G12.RD} := \frac{\omega_1 \cdot b_{eff.t.wc} \cdot t_{w.c} \cdot f_{y.c}}{\gamma_{M0}} = 1804.29\ kN$$

b) Stiffness:

$$k_{3.1} := \frac{0.7 \cdot b_{eff.t.wc} \cdot t_{w.c}}{d_c} = 17.09\ mm \qquad k_{3.2} := k_{3.1}$$

4.- Column Flange in Bending (4):

a) Resistance Stregth:

$$F_{t.bo} := \frac{0.9 \cdot f_{u.bo} \cdot A_s}{\gamma_{M2}} = 254.16\ kN$$

Tension Resistante of a single Bolt.

Effective Lengths at Column Flange:

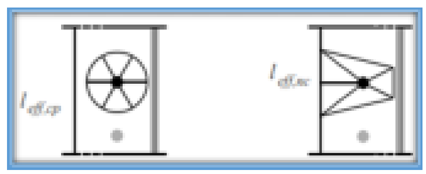
Table 6.4: Effective lengths for an unstiffened column flange

Bolt-row Location	Bolt-row considered individually		Bolt-row considered as part of a group of bolt-rows	
	Circular patterns $\ell_{eff,cp}$	Non $\ell_{eff,nc}$	Circular patterns $\ell_{eff,cp}$	Non-circular patterns $\ell_{eff,nc}$
Inner bolt-row	$2\pi m$	$4m + 1{,}25e$	$2p$	p
End bolt-row	The smaller of: $2\pi m$; $\pi m + 2e_1$	The smaller of: $4m + 1{,}25e$; $2m + 0{,}625e + e_1$	The smaller of: $\pi m + p$; $2e_1 + p$	The smaller of: $2m + 0{,}625e + 0{,}5p$; $e_1 + 0{,}5p$
Mode 1:	$\ell_{eff,1} = \ell_{eff,nc}$ but $\ell_{eff,1} \le \ell_{eff,cp}$		$\sum \ell_{eff,1} = \sum \ell_{eff,nc}$ but $\sum \ell_{eff,1} \le \sum \ell_{eff,cp}$	
Mode 2:	$\ell_{eff,2} = \ell_{eff,nc}$		$\sum \ell_{eff,2} = \sum \ell_{eff,nc}$	

e_1 is the distance from the centre of the fasteners in the end row to the adjacent free end of the column flange measured in the direction of the axis of the column profile (see row 1 and row 2 in Figure 6.9).

Row 1:

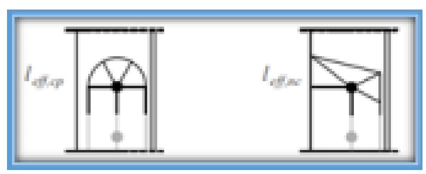
As Individual



$$l_{eff.I.cp.R1} := 2 \cdot \pi \cdot m_{fc} = 112.47\ mm$$

$$l_{eff.I.nc.R1} := 4 \cdot m_{fc} + 1.25 \cdot e_{fc} = 200.98\ mm$$

As First Bolt Row of Group 1-2



$$l_{eff.G12.cp.R1} := 2 \cdot p_1 = 400.00\ mm$$

$$l_{eff.G12.nc.R1} := p_1 = 200.00\ mm$$

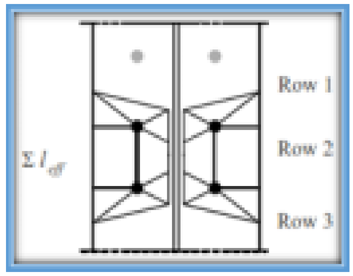
Row 2:

As Individual

$$l_{eff.I.cp.R2} := l_{eff.I.cp.R1} = 112.47\ mm$$

$$l_{eff.I.nc.R2} := l_{eff.I.nc.R1} = 200.98\ mm$$

As End Bolt Row of Group 1-2



$l_{eff.G12.cp.R2} := l_{eff.G12.cp.R1} = 400.00\ mm$

$l_{eff.G12.nc.R2} := l_{eff.G12.nc.R1} = 200.00\ mm$

T-Sub Resistances at Column Flange:

Individual Bolt Row 1 Resistance

Mode 1:

$l_{eff.I.R1} := min\left(l_{eff.I.cp.R1}, l_{eff.I.nc.R1}\right) = 112.47\ mm$

$$M_{pl1.R1.fc.RD} := \frac{l_{eff.I.R1} \cdot t_{f.c}^{\ 2} \cdot f_{y.c}}{4 \cdot \gamma_{M0}} = 12.37\ kN \cdot m$$

$$F_{t1.I.R1.fc.RD} := \frac{4 \cdot M_{pl1.R1.fc.RD}}{m_{fc}} = 2764.60\ kN$$

Mode 2:

$$M_{pl2.R1.fc.RD} := \frac{l_{eff.I.nc.R1} \cdot t_{f.c}^{\ 2} \cdot f_{y.c}}{4 \cdot \gamma_{M0}} = 22.11\ kN \cdot m$$

$$F_{t2.I.R1.fc.RD} := \frac{2 \cdot M_{pl2.R1.fc.RD} + n_{fc} \cdot \left(n_1 \cdot F_{t.bo}\right)}{m_{fc} + n_{fc}} = 1114.81\ kN$$

Mode 3:

$F_{t3.I.R1.fc.RD} := n_1 \cdot F_{t.bo} = 508.32\ kN$

$F_{4.I.R1.RD} := min\left(F_{t1.I.R1.fc.RD}, F_{t2.I.R1.fc.RD}, F_{t3.I.R1.fc.RD}\right) = 508.32\ kN$

if $F_{4.I.R1.RD} = F_{t1.I.R1.fc.RD}$
  "Brittle Failure"
else
  "OK"
= "Brittle Failure"

Individual Bolt Row 1 Resistance

$F_{4.I.R2.RD} := F_{4.I.R1.RD} = 508.32\ kN$

Resistance Bolt Group 1-2

Mode 1:

$$l_{eff.G12.R1} := min\left(l_{eff.G12.cp.R1}, l_{eff.G12.nc.R1}\right) = 200.00\ mm$$

$$M_{pl.1.fc.RD} := \frac{\left(2 \cdot l_{eff.G12.R1}\right) \cdot t_{f.c}^{\ 2} \cdot f_{y.c}}{4 \cdot \gamma_{M0}} = 44000.00\ kN \cdot mm$$

$$F_{t.G12.1.fc.RD} := \frac{4 \cdot M_{pl.1.fc.RD}}{m_{fc}} = 9832.40\ kN$$

Mode 2:

$$M_{pl.2.fc.RD} := \frac{\left(2 \cdot l_{eff.G12.nc.R1}\right) \cdot t_{f.c}^{\ 2} \cdot f_{y.c}}{4 \cdot \gamma_{M0}} = 44000.00\ kN \cdot mm$$

$$F_{t.G12.2.fc.RD} := \frac{2 \cdot M_{pl.2.fc.RD} + n_{fc} \cdot \left(n_{t.bo} \cdot F_{t.bo}\right)}{m_{fc} + n_{fc}} = 2222.20\ kN$$

Mode 3:

$$F_{t.G12.3.fc.RD} := n_{t.bo} \cdot F_{t.bo} = 1016.64\ kN$$

$$F_{4.G12.RD} := min\left(F_{t.G12.1.fc.RD}, F_{t.G12.2.fc.RD}, F_{t.G12.3.fc.RD}\right) = 1016.64\ kN$$

if $F_{t.G12.RD} = F_{t.G12.3.fc.RD}$ = "Brittle Failure"
  "Brittle Failure"
else
  "OK"

b) Stiffness:

$$k_{4.1} := \frac{0.9 \cdot l_{eff.I.R1} \cdot t_{f.c}^{\ 3}}{m_{fc}^{\ 3}} = 1129.53\ mm$$

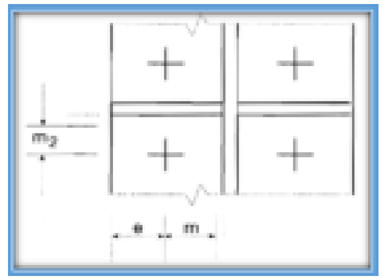
$$k_{4.2} := \frac{0.9 \cdot l_{eff.I.R1} \cdot t_{f.c}^{\ 3}}{m_{fc}^{\ 3}} = 1129.53\ mm$$

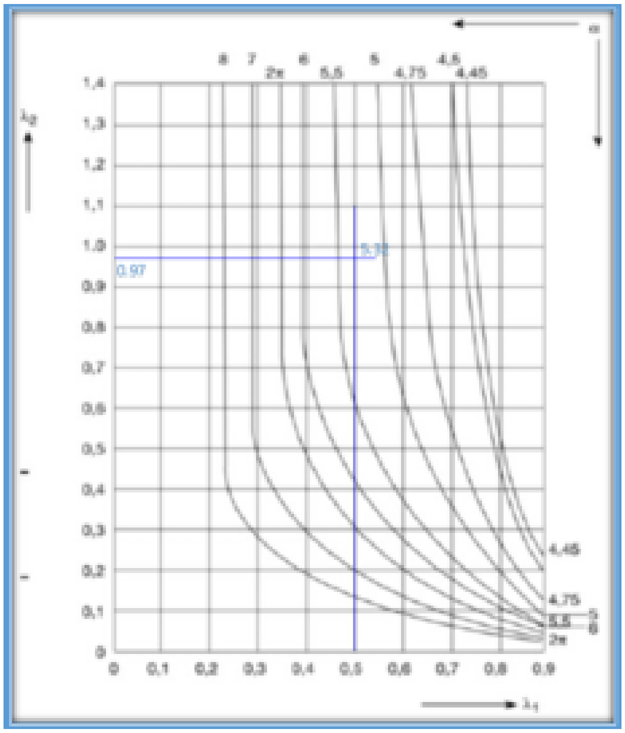
$$k_{4.1\_2} := \frac{0.9 \cdot \left(2 \cdot l_{eff.G12.R1}\right) \cdot t_{f.c}^{\ 3}}{m_{fc}^{\ 3}} = 4017.20\ mm$$

5.- End Plate in Bending (5):

a) Resistance Stregth:

$$\lambda_1 := \frac{m_p}{m_p + e_p} = 0.50 \qquad \lambda_2 := \frac{m_{2.p}}{m_p + e_p} = 0.97$$





$$\alpha_p := 5.32$$

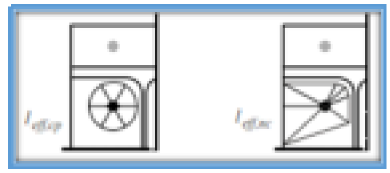
Effective Lengths at Column Flange:

Table 6.6: Effective lengths for an end-plate

Bolt-row location	Bolt-row considered individually		Bolt-row considered as part of a group of bolt-rows	
	Circular patterns $\ell_{eff,cp}$	Non-circular patterns $\ell_{eff,nc}$	Circular patterns $\ell_{eff,cp}$	Non-circular patterns $\ell_{eff,nc}$
Bolt-row outside tension flange of beam	Smallest of: $2\pi m_x$, $\pi m_x + w$, $\pi m_x + 2e$	Smallest of: $4m_x + 1,25e_x$, $e+2m_x+0,625e_x$, $0,5b_p$, $0,5w+2m_x+0,625e_x$	—	—
First bolt-row below tension flange of beam	$2\pi m$	αm	$\pi m + p$	$0,5p + \alpha m - (2m + 0,625e)$
Other inner bolt-row	$2\pi m$	$4m + 1,25\,e$	$2p$	p
Other end bolt-row	$2\pi m$	$4m + 1,25\,e$	$\pi m + p$	$2m+0,625e+0,5p$
Mode 1:	$\ell_{eff,1} = \ell_{eff,nc}$ but $\ell_{eff,1} \le \ell_{eff,cp}$		$\sum \ell_{eff,1} = \sum \ell_{eff,nc}$ but $\sum \ell_{eff,1} \le \sum \ell_{eff,cp}$	
Mode 2:	$\ell_{eff,2} = \ell_{eff,nc}$		$\sum \ell_{eff,2} = \sum \ell_{eff,nc}$	
α should be obtained from Figure 6.11.				

Row 1:

As Individual



$$l_{eff.I.cp.R1} := 2\ \pi \cdot m_p = 251.60\ mm$$

$$l_{eff.I.nc.R1} := \alpha_p \cdot m_p = 213.03\ mm$$

As First Row of Bolt Group 1-2

$$l_{eff.G12.cp.R1} := \pi \cdot m_p + p_1 = 325.80\ mm$$

$$l_{eff.G12.nc.R1} := 0.5 \cdot p_1 + \alpha_p \cdot m_p - \left(2 \cdot m_p + 0.625 \cdot e_p\right) = 207.94\ mm$$

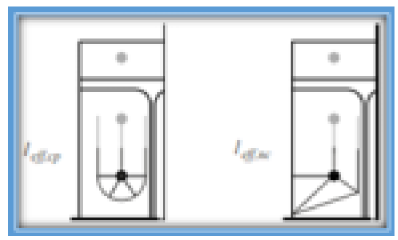
Row 2:

As Individual

$$l_{eff.I.cp.R2} := l_{eff.I.cp.R1} = 251.60\ mm$$

$$l_{eff.I.nc.R2} := l_{eff.I.nc.R1} = 213.03\ mm$$

As End Row of Bolt Group 1-2



$$l_{eff.G12.cp.R2} := \pi \cdot m_p + p_1 = 325.80\ mm$$

$$l_{eff.G12.nc.R2} := 0.5 \cdot p_1 + 2 \cdot m_p + 0.625 \cdot e_p = 205.09\ mm$$

T-Sub Resistances at End Plate:

Individual Bolt Row 1 Resistance

Mode 1:

$$l_{eff.I.R1} := min\left(l_{eff.I.nc.R1}, l_{eff.I.cp.R1}\right) = 213.03\ mm$$

$$M_{pl.1.p.RD} := \frac{l_{eff.I.R1} \cdot t_p^{\ 2} \cdot f_{y.p}}{4 \cdot \gamma_{M0}} = 1565.77\ kN \cdot mm$$

$$F_{t.I1.1.p.RD} := \frac{4 \cdot M_{pl.1.p.RD}}{m_p} = 156.41\ kN$$

Mode 2:

$$M_{pl.2.p.RD} := \frac{l_{eff.I.nc.R1} \cdot t_p^{\ 2} \cdot f_{y.p}}{4 \cdot \gamma_{M0}} = 1565.77\ kN \cdot mm$$

$$F_{t.I1.2.p.RD} := \frac{2 \cdot M_{pl.2.p.RD} + n_p \cdot n_1 \cdot F_{t.bo}}{m_p + n_p} = 293.15\ kN$$

Mode 3:

$$F_{t.I1.3.p.RD} := n_1 \cdot F_{t.bo} = 508.32\ kN$$

$$F_{5.I.R1.RD} := min\left(F_{t.I1.1.p.RD}, F_{t.I1.2.p.RD}, F_{t.I1.3.p.RD}\right) = 156.41\ kN$$

if $F_{5.I.R1.RD} = F_{t.I1.3.p.RD}$ = "OK"
"Brittle Failure"
else
"OK"

Individual Bolt Row 2 Resistance

$$F_{5.I.R2.RD} := F_{5.I.R1.RD} = 156.41\ kN$$

Group of Bolts 1-2 Resistance

Mode 1:

$$l_{eff.G12} := min\left(l_{eff.G12.cp.R1} + l_{eff.G12.cp.R2}, l_{eff.G12.nc.R1} + l_{eff.G12.nc.R2}\right) = 413.03\ mm$$

$$M_{pl.1.p.RD} := \frac{l_{eff.G12} \cdot t_p^{\ 2} \cdot f_{y.p}}{4 \cdot \gamma_{M0}} = 3035.77\ kN \cdot mm$$

$$F_{t.G12.1.p.RD} := \frac{4 \cdot M_{pl.1.p.RD}}{m_p} = 303.25\ kN$$

Mode 2:

$$M_{pl.2.p.RD} := \frac{\left(l_{eff.G12.nc.R1} + l_{eff.G12.nc.R2}\right) \cdot t_p^{\ 2} \cdot f_{y.p}}{4 \cdot \gamma_{M0}} = 3035.77\ kN \cdot mm$$

$$F_{t.G12.2.p.RD} := \frac{2 \cdot M_{pl.2.p.RD} + n_p \cdot n_{t.bo} \cdot F_{t.bo}}{m_p + n_p} = 583.90\ kN$$

Mode 3:

$$F_{t.G12.3.p.RD} := n_{t.bo} \cdot F_{t.bo} = 1016.64\ kN$$

$$F_{5.G12.RD} := min\left(F_{t.G12.1.p.RD}, F_{t.G12.2.p.RD}, F_{t.G12.3.p.RD}\right) = 303.25\ kN$$

$$\begin{array}{l} \text{if } F_{5.G12.RD} = F_{t.G12.3.p.RD} \\ \quad \| \text{“Brittle Failure”} \\ \text{else} \\ \quad \| \text{“OK”} \end{array} = \text{“OK”}$$

b) Stiffness:

$$l_{eff.k.R1} := min\left(l_{eff.I.cp.R1}, l_{eff.I.nc.R1}, l_{eff.G12.cp.R1}, l_{eff.G12.nc.R1}\right) = 207.94\ mm$$

$$k_{5.1} := \frac{0.9 \cdot l_{eff.k.R1} \cdot t_p^{\ 3}}{m_p^{\ 3}} = 2.91\ mm$$

$$l_{eff.k.R2} := min\left(l_{eff.I.cp.R2}, l_{eff.I.nc.R2}, l_{eff.G12.cp.R2}, l_{eff.G12.nc.R2}\right) = 205.09\ mm$$

$$k_{5.2} := \frac{0.9 \cdot l_{eff.k.R2} \cdot t_p^{\ 3}}{m_p^{\ 3}} = 2.87\ mm$$

6.- Beam Flange and Web in Compression (7):

a) Resistance Stregth:

$$\begin{array}{l} \text{if } \dfrac{b_{f.b} - \left(t_{w.b} + 2 \cdot r_b\right)}{2 \cdot t_{f.c}} \le 9 \cdot \varepsilon_b \\ \quad \| \text{“Flange Class 1”} \\ \text{else} \\ \quad \left\| \begin{array}{l} \text{if } \dfrac{b_{f.b} - \left(t_{w.b} + 2 \cdot r_b\right)}{2 \cdot t_{f.c}} \le 10 \cdot \varepsilon_b \\ \quad \| \text{“Flange Class 2”} \\ \text{else} \\ \quad \| \text{“Flange Class 3”} \end{array} \right. \end{array} = \text{“Flange Class 1”}$$

$$\begin{array}{l} \text{if } \dfrac{d_c}{t_{w.c}} \le 72 \cdot \varepsilon_c \\ \quad \| \text{“Web Class 1”} \\ \text{else} \\ \quad \left\| \begin{array}{l} \text{if } \dfrac{d_c}{t_{w.c}} \le 83 \cdot \varepsilon_c \\ \quad \| \text{“Web Class 2”} \\ \text{else} \\ \quad \| \text{“Web Class 3”} \end{array} \right. \end{array} = \text{“Web Class 1”}$$

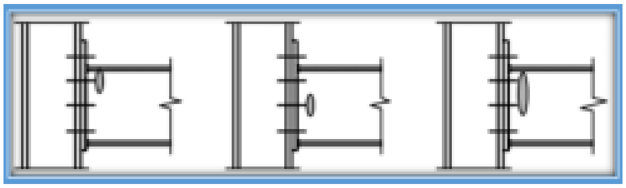
$$M_{c.RD} := \frac{W_{pl.b} \cdot f_{y.b}}{\gamma_{M0}} = 359.43\ kN \cdot m$$

$$F_{7.RD} := \frac{M_{c.RD}}{h_b - t_{f.b}} = 929.95\ kN$$

b) Stiffness:

$$k_7 := \infty$$

7.- Beam Web in Tension (8):



a) Resistance Stregth:

Individual Bolt Row 1 Resistance

$$F_{8.I.R1.RD} := \frac{l_{eff.I.R1} \cdot t_{w.b} \cdot f_{y.b}}{\gamma_{M0}} = 503.81\ kN$$

Individual Bolt Row 2 Resistance

$$F_{8.I.R2.RD} := F_{8.I.R1.RD} = 503.81\ kN$$

Group of Bolts 1-2 Resistance

$$F_{8.G12.RD} := \frac{l_{eff.G12} \cdot t_{w.b} \cdot f_{y.b}}{\gamma_{M0}} = 976.81\ kN$$

b) Stiffness:

$$k_8 := \infty$$

8.- Bolts in Tension (10):

a) Resistance Stregth:

$$F_{10.RD} := n_{t.bo} \cdot F_{t.bo} = 1016.64\ kN$$

b) Stiffness:

$$L_{bo} := (t_{f.c} + t_p) + \left(\frac{t_{h.bo} + t_{w.bo}}{2} \right) + t_{wa} = 70.00\ mm$$

$$k_{10} := \frac{1.6 \cdot A_s}{L_{bo}} = 8.07\ mm$$

9.- Welds (19):

a) Resistance Stregth:

$$F_{19.RD} := \frac{f_{u.p} \cdot a_f}{\sqrt{2} \cdot \beta_w \cdot \gamma_{M2}} \cdot \left(b_{f.b} \cdot 2 - t_{w.b} - 2 \cdot r_b\right) + \frac{f_{u.p} \cdot a_w}{\sqrt{2} \cdot \beta_w \cdot \gamma_{M2}} \cdot \left(\frac{d_b}{2}\right) = 569.85\ kN$$

$$M_{j.ED} := F_{19.RD} \cdot \left(h_p - 7\ mm\right) = 215.97\ kN \cdot m$$

Maximum Applied Moment due to Weld Resistance

b) Stiffness:

$$k_{19} := \infty$$

Resistance and Stiffness Assembly:

Moment Resistance of the Joint ($M_{j.RD}$):

Individual Resistance of Bolt Rows (Tension):

$$F_{I.R1.RD} := min\left(F_{3.I.RD}, F_{4.I.R1.RD}, F_{5.I.R1.RD}\right) = 156.41\ kN$$

$$F_{I.R2.RD} := min\left(F_{3.I.RD}, F_{4.I.R2.RD}, F_{5.I.R2.RD}\right) = 156.41\ kN$$

Group 1-2 Bolt Resistance (Tension)

$$F_{G12.\Sigma I.RD} := F_{I.R1.RD} + F_{I.R2.RD} = 312.82\ kN$$

$$F_{G12.RD} := min\left(F_{G12.\Sigma I.RD}, F_{3.G12.RD}, F_{4.G12.RD}\right) = 312.82\ kN$$

if $F_{G12.\Sigma I.RD} \leq F_{G12.RD}$
  "Individual Resistance is Reached"
else
  "Consider Group Effect"
= "Individual Resistance is Reached"

Maximum Tension Load per Row:

$$F_{T.R1.RD} := min\left(F_{I.R1.RD}, F_{8.G12.RD}, F_{10.RD}, F_{19.RD}\right) = 156.41\ kN$$

$$F_{T.R2.RD} := min\left(F_{I.R2.RD}, F_{8.G12.RD}, F_{10.RD}, F_{19.RD}\right) = 156.41\ kN$$

Maximum Load Resisted by other Components:

$$F_{T.glob.RD} := min\left(F_{1.RD}, F_{2.RD}, F_{7.RD}\right) = 929.95\ kN$$

$$\begin{array}{l|l} \text{if } \left(F_{T.R1.RD} + F_{T.R2.RD}\right) \le F_{T.glob.RD} & = \text{“Individual Resistance is Reached”} \\ \quad \Vert \text{“Individual Resistance is Reached”} & \\ \text{else} & \\ \quad \Vert \text{“Consider Global Effect”} & \end{array}$$

Moment Resistance:

$$M_{j.RD} := F_{T.R1.RD} \cdot z + F_{T.R2.RD} \cdot z_2 = 60.45\ kN \cdot m$$

Design Stiffness of the Joint ($S_{j.ini.RD}$,$S_{j.RD}$):

Initial Stiffness (Elastic):

$$k_{eff.1} := \frac{1}{\frac{1}{k_{3.1}} + \frac{1}{k_{4.1}} + \frac{1}{k_{5.1}} + \frac{1}{k_{10}}} = 1.90\ mm \qquad k_{eff.2} := \frac{1}{\frac{1}{k_{3.2}} + \frac{1}{k_{4.2}} + \frac{1}{k_{5.2}} + \frac{1}{k_{10}}} = 1.88\ mm$$

$$z_{eq} := \frac{k_{eff.1} \cdot z^2 + k_{eff.2} \cdot {z_2}^2}{k_{eff.1} \cdot z + k_{eff.2} \cdot z} = 162.05\ mm \qquad k_{eq} := \frac{k_{eff.1} \cdot z + k_{eff.2} \cdot z_2}{z_{eq}} = 4.52\ mm$$

$$S_{j.ini} := \frac{E \cdot z^2}{\frac{1}{k_1} + \frac{1}{k_2} + \frac{1}{k_{eq}}} = 52.15\ MN \cdot m$$

Secant Stiffness (Non-linear):

$$\psi := 2.7 \qquad M_{j.ED} := 200\ kN \cdot m$$

$$\begin{array}{l|l} \mu := \text{if } M_{j.ED} \le \frac{2}{3} \cdot M_{j.RD} & = 75.58 \\ \quad \Vert\ 1 & \\ \quad \text{else} & \\ \quad \Vert \left(1.5 \cdot \frac{M_{j.ED}}{M_{j.RD}}\right)^{\psi} & \end{array} \qquad S_{j.RD} := \frac{S_{j.ini}}{\mu} = 0.69\ MN \cdot m$$

C.2 Rigid Joint

Imput Parameters:

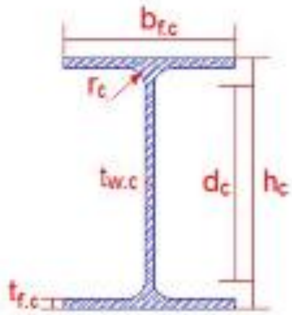
$\gamma_{M0} := 1.0$ $\gamma_{Mb} := 1.00$ $E := 210000 \dfrac{N}{mm^2}$

$\gamma_{M1} := 1.1$ $\gamma_{Mw} := 1.00$ $\beta := 1.0$

$\gamma_{M2} := 1.25$ Steel: S235

Section Properties and Geometry:

Column (HEM 200):



$h_c := 222\ mm$ $r_c := 18\ mm$ $f_{y.cfl} := 267\ MPa$

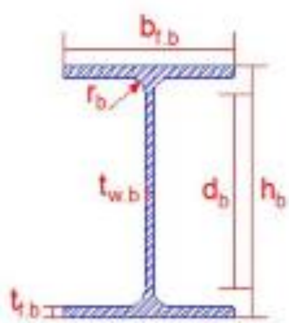
$b_{f.c} := 204\ mm$ $d_c := 134.8\ mm$

$t_{w.c} := 15.9\ mm$ $f_{y.cw} := 324\ MPa$ $\varepsilon_c := \sqrt{\dfrac{235\ MPa}{f_{y.cw}}} = 0.85$

$t_{f.c} := 25.6\ mm$ $f_{u.c} := 360\ MPa$

$A_c := 2 \cdot b_{f.c} \cdot t_{f.c} + \left(h_c - 2 \cdot t_{f.c}\right) \cdot t_{w.c} + 0.858\ r_c^{\ 2} = 134.39\ cm^2$

Beamn (IPE 500):



$h_b := 498\ mm$ $r_b := 21\ mm$ $I_b := 48200\ cm^4$

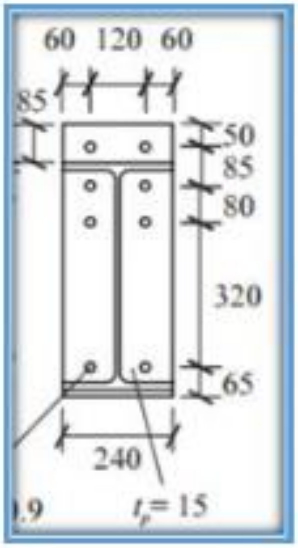
$b_{f.b} := 199\ mm$ $d_b := 425.8\ mm$ $f_{y.bfl} := 248\ MPa$

$t_{w.b} := 10.5\ mm$ $f_{y.bw} := 281\ MPa$ $\varepsilon_b := \sqrt{\dfrac{235\ MPa}{f_{y.bw}}} = 0.91$

$t_{f.b} := 15.1\ mm$ $f_{u.b} := 360\ MPa$ $W_{pl.b} := 2194\ cm^3$

$A_b := 2 \cdot b_{f.b} \cdot t_{f.b} + \left(h_b - 2 \cdot t_{f.b}\right) \cdot t_{w.b} + 0.858\ r_b^{\ 2} = 113.00\ cm^2$

Joint Geometry:



Welds:

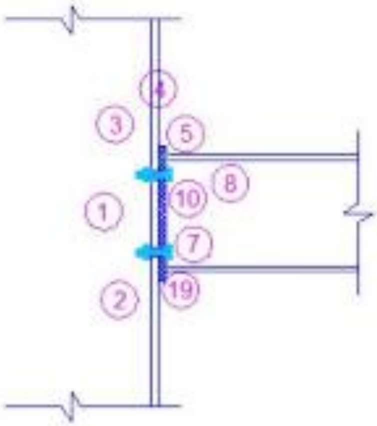
$a_{fl} := 14\ mm$

$a_w := 0\ mm$

$\beta_w := 0.9$

$z := h_b - t_{f.b} = 482.9\ mm$

Calculation of the Individual Componets:



1 - Column Web Panel in Shear
2 - Column Web in Transverse Compression
3 - Column Web in Tension
4 - Column Flange in Bending
7 - Beam Web/Flange in Compresion
8 - Beam Web in Tension
19 - Welds

Calculation of the Individual Componets:

1.- Column Web Panel in Shear (1):

a) Resistance:

$A_{v.c} := A_c - 2 \cdot b_{f.c} \cdot t_{f.c} + \left(t_{w.c} + 2 \cdot r_c\right) \cdot t_{f.c} = 4322\ mm^2$

$$V_{wp.RD} := \frac{0.9 \cdot f_{y.cw} \cdot A_{v.c}}{\sqrt{3} \cdot \gamma_{M0}} = 727.69\ kN$$

$$F_{1.RD} := \frac{V_{wp.RD}}{\beta} = 727.69\ kN$$

Limitation:

$$\text{if } \frac{d_c}{t_{w.c}} \le 69 \cdot \sqrt{\frac{235\ MPa}{f_{y.cw}}} \;\; \| \text{"OK"} \;\; \text{else} \;\; \| \text{"Web Slender too High"} \;\; = \text{"OK"}$$

b) Stiffness:

$$k_1 := \frac{0.38 \cdot A_{v.c}}{\beta \cdot z} = 3.40\ mm$$

2.- Column Web in Transverse Compression (2):

a) Resistance Stregth:

$$s := r_c \qquad b_{eff.c.wc} := t_{f.b} + 2 \cdot \sqrt{2} \cdot a_{fl} + 5 \cdot (t_{f.c} + s) = 272.70\ mm$$

$$\omega_1 := \frac{1}{\sqrt{1 + 1.3 \left(\frac{b_{eff.c.wc} \cdot t_{w.c}}{A_{v.c}}\right)^2}} = 0.658 \qquad k_{w.c} := 1$$

$$\lambda_p := 0.932 \cdot \sqrt{\frac{b_{eff.c.wc} \cdot d_c \cdot f_{y.cw}}{E \cdot t_{w.c}^{\,2}}} = 0.441 \qquad \rho := \text{if } \lambda_p > 0.72 \;\; \left\| \frac{(\lambda_p - 0.2)}{\lambda_p^{\,2}} \right. \;\; \text{else} \;\; \| 1 \;\; = 1.000$$

$$F_{2.RD} := min\left(\frac{\omega_1 \cdot k_{w.c} \cdot b_{eff.c.wc} \cdot t_{w.c} \cdot f_{y.cw}}{\gamma_{M0}}, \frac{\omega_1 \cdot k_{w.c} \cdot \rho \cdot b_{eff.c.wc} \cdot t_{w.c} \cdot f_{y.cw}}{\gamma_{M1}}\right) = 840.62\ kN$$

b) Stiffness:

$$k_2 := \frac{0.7 \cdot b_{eff.c.wc} \cdot t_{w.c}}{d_c} = 22.52\ mm$$

3.- Column Web in Transverse Tension (3):

a) Resistance Stregth:

$s := r_c$ $\qquad b_{eff.t.wc} := t_{f.b} + 2 \cdot \sqrt{2} \cdot a_{fl} + 5 \cdot (t_{f.c} + s) = 272.70\ mm$

$$\omega_1 := \frac{1}{\sqrt{1 + 1.3 \left(\frac{b_{eff.t.wc} \cdot t_{w.c}}{A_{v.c}}\right)^2}} = 0.658$$

$$\lambda_p := 0.932 \cdot \sqrt{\frac{b_{eff.t.wc} \cdot d_c \cdot f_{y.cw}}{E \cdot t_{w.c}^{\ 2}}} = 0.441 \qquad \rho := \begin{array}{l} \text{if } \lambda_p > 0.72 \\ \quad \frac{(\lambda_p - 0.2)}{\lambda_p^{\ 2}} \\ \text{else} \\ \quad 1 \end{array} = 1.000$$

$$F_{3.RD} := \frac{\omega_1 \cdot b_{eff.c.wc} \cdot t_{w.c} \cdot f_{y.cw}}{\gamma_{M0}} = 924.68\ kN$$

b) Stiffness:

$$k_3 := \frac{0.7 \cdot b_{eff.t.wc} \cdot t_{w.c}}{d_c} = 22.52\ mm$$

4.- Column Flange in Bending (4):

a) Resistance Stregth:

$$k := min\left(\frac{t_{f.c}}{t_{f.b}} \cdot \frac{f_{y.cfl}}{f_{y.bfl}}, 1\right) = 1.00$$

$$b_{eff.b.cf} := t_{w.c} + 2 \cdot s + 7 \cdot k \cdot t_{f.c} = 231.10\ mm$$

$$F_{4.RD} := b_{eff.b.cf} \cdot t_{f.b} \cdot f_{y.bfl} = 865.42\ kN$$

b) Stiffness:

$$k_4 := \infty$$

6.- Beam Flange and Web in Compression (7):

a) Resistance Stregth:

$$M_{c.RD} := \frac{W_{pl.b} \cdot f_{y.bfl}}{\gamma_{M0}} = 544.11\ kN \cdot m$$

$$F_{7.RD} := \frac{M_{c.RD}}{z} = 1126.76\ kN$$

b) Stiffness:

$$k_7 := \infty$$

7.- Beam Web in Tension (8):

a) Resistance Stregth:

$$b_{eff.t.wb} := b_{eff.t.wc} = 272.70\ mm$$

$$F_{8.RD} := \frac{b_{eff.t.wb} \cdot t_{w.b} \cdot f_{y.bw}}{\gamma_{M0}} = 804.60\ kN$$

b) Stiffness:

$$k_8 := \infty$$

9.- Welds (19):

a) Resistance Stregth:

$$l_{weld} := 2 \cdot b_{f.b} - \left(t_{w.b} + 2 \cdot r_b\right) = 345.50\ mm$$

$$F_{19.afl.RD} := \frac{f_{u.b} \cdot a_{fl}}{\sqrt{2} \cdot \beta_w \cdot \gamma_{M2}} \cdot l_{weld} = 1094.49\ kN$$

$$M_{j.ED} := F_{19.afl.RD} \cdot \left(h_b - t_{f.b}\right) = 528.53\ kN \cdot m$$

Maximum Applied Moment due to Weld Resistance

b) Stiffness:

$$k_{19} := \infty$$

Resistance and Stiffness Assembly:

Moment Resistance of the Joint ($M_{j.RD}$):

Maximum Tension Load

$$F_{T.RD} := min\left(F_{3.RD}, F_{4.RD}, F_{8.RD}, F_{19.afl.RD}\right) = 804.60\ kN$$

Maximum Load Resisted by other Components:

$$F_{T.glob.RD} := min\left(F_{1.RD}, F_{2.RD}, F_{7.RD}\right) = 727.69\ kN$$

$$\begin{array}{l} \text{if } F_{T.RD} \le F_{T.glob.RD} \\ \quad \| \text{“Tension Resistance is Reached”} \\ \text{else} \\ \quad \| \text{“Consider Compression Resistance”} \end{array} = \text{“Consider Compression Resistance”}$$

Moment Resistance:

$$M_{j.RD} := F_{T.glob.RD} \cdot z = 351.40\ kN \cdot m$$

Design Stiffness of the Joint ($S_{j.RD}$):

Initial Stiffness (Elastic):

$$S_{j.ini} := \frac{E \cdot z^2}{\frac{1}{k_1} + \frac{1}{k_2} + \frac{1}{k_3}} = 127.92\ MN \cdot m$$

Secant Stiffness (Non-linear):

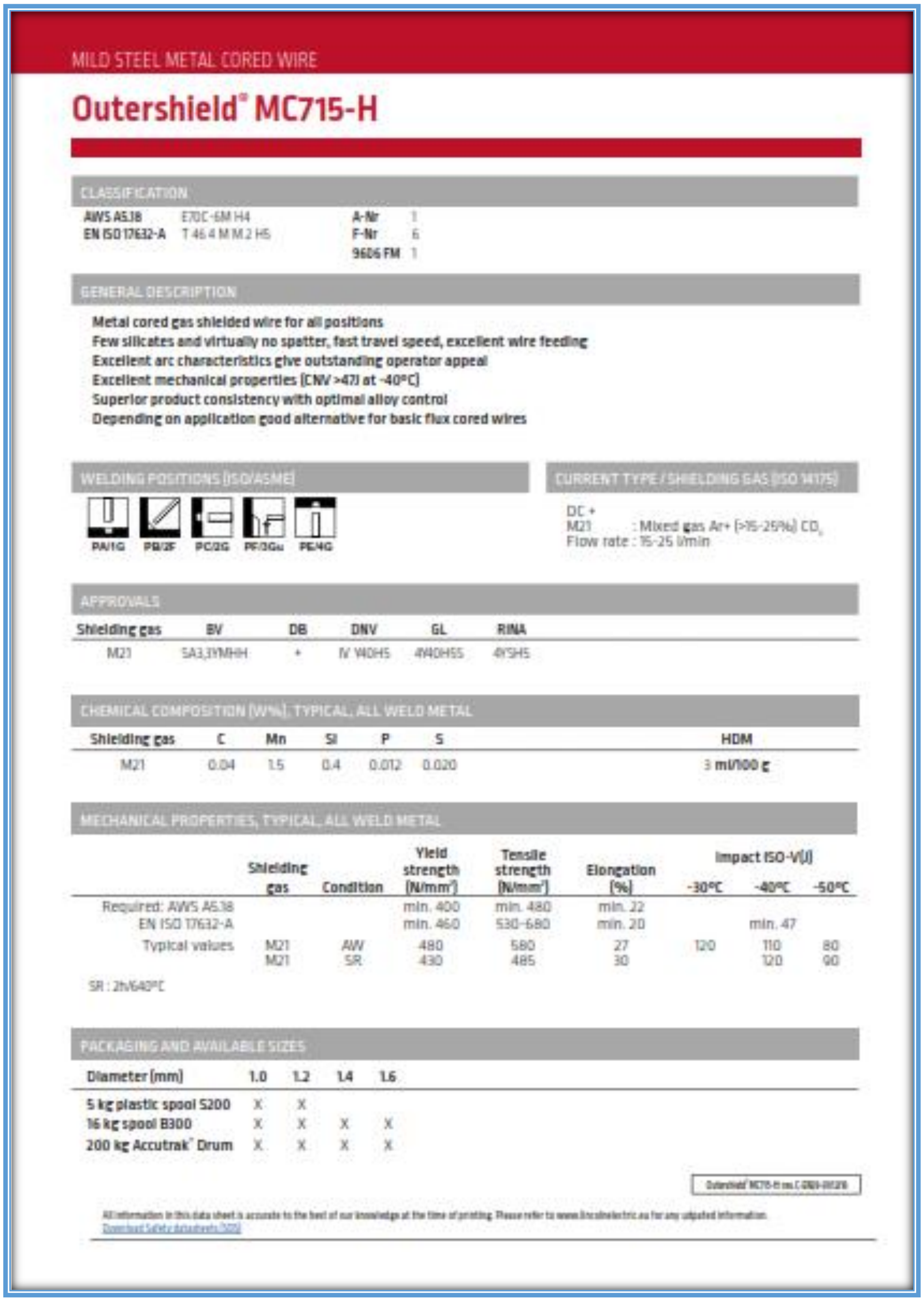
$\psi := 2.7 \qquad M_{j.ED} := 250\ kN \cdot m$

$$\mu := \begin{array}{l} \text{if } M_{j.ED} \le \frac{2}{3} \cdot M_{j.RD} \\ \quad \| 1 \\ \text{else} \\ \quad \| \left(1.5 \cdot \frac{M_{j.ED}}{M_{j.RD}}\right)^{\psi} \end{array} = 1.19$$

$$S_{j.RD} := \frac{S_{j.ini}}{\mu} = 107.33\ MN \cdot m$$

Annex D – Filler Material Properties

D.1 Filler Material S355



MILD STEEL METAL CORED WIRE

Outershield® MC715-H

CLASSIFICATION

AWS A5.18	E70C-6M H4	**A-Nr**	1
EN ISO 17632-A	T 46 4 M M 2 H5	**F-Nr**	6
		9606 FM	1

GENERAL DESCRIPTION

- **Metal cored gas shielded wire for all positions**
- **Few silicates and virtually no spatter, fast travel speed, excellent wire feeding**
- **Excellent arc characteristics give outstanding operator appeal**
- **Excellent mechanical properties (CNV >47J at -40°C)**
- **Superior product consistency with optimal alloy control**
- **Depending on application good alternative for basic flux cored wires**

WELDING POSITIONS (ISO/ASME)

PA/1G PB/2F PC/2G PF/3Gu PE/4G

CURRENT TYPE / SHIELDING GAS (ISO 14175)

DC +
M21 : Mixed gas Ar+ (>15-25%) CO_2
Flow rate : 15-25 l/min

APPROVALS

Shielding gas	BV	DB	DNV	GL	RINA
M21	SA3,3YMHH	+	IV Y40H5	4Y40H55	4YSH5

CHEMICAL COMPOSITION (W%), TYPICAL, ALL WELD METAL

Shielding gas	C	Mn	Si	P	S	HDM
M21	0.04	1.5	0.4	0.012	0.020	3 ml/100 g

MECHANICAL PROPERTIES, TYPICAL, ALL WELD METAL

	Shielding gas	Condition	Yield strength (N/mm²)	Tensile strength (N/mm²)	Elongation (%)	Impact ISO-V(J) -30°C	Impact ISO-V(J) -40°C	Impact ISO-V(J) -50°C
Required: AWS A5.18			min. 400	min. 480	min. 22			
EN ISO 17632-A			min. 460	530-680	min. 20		min. 47	
Typical values	M21	AW	480	580	27	120	110	80
	M21	SR	430	485	30		120	90

SR : 2h/640°C

PACKAGING AND AVAILABLE SIZES

Diameter (mm)	1.0	1.2	1.4	1.6
5 kg plastic spool S200	X	X		
16 kg spool B300	X	X	X	X
200 kg Accutrak® Drum	X	X	X	X

All information in this data sheet is accurate to the best of our knowledge at the time of printing. Please refer to www.lincolnelectric.eu for any updated information.
Download Safety datasheets (SDS)

Figure D - 1: S355 Typical Filler Material

D.2 Filler Material S690

CEWELD®

Megafil 742M

CATEGORIE	FCAW Flux-Cored
TYPE	Seamless high strength metal-cored wire for M21 without slag.
APPLICATIONS	Crane-, plant-, craft-, lifting and steel construction, pipe work, foundries.
PROPERTIES	Remarkable crack resistant weld metal in combination with very low (<3ml/100gr) hydrogen content. Therefore, suitable for the economic processing of high-strength and low temperature fine grained structural steels. Excellent welding properties in short and spray arc. High deposition rate and no intermediate cleaning required with very low spatter loss. Excellent wetting properties compare to solid wires that results in a bigger parameter range and improved duty cycle for the welder.
CLASSIFICATION	AWS A5.28: E110C-K4 H4 A5.36: E111T15-M21A8-K4-H4 EN ISO 18276-A: T 69 6 Mn2NiCrMo M M21 1 H5
SUITABLE FOR	Naxtra 70, Weldox 700, S690, S620, EStE 690, 690V, XABO 620, S620QI1, S690QL1, S600MC, S700MC, Naxtra 63, Naxtra 70, Optim 700 mc plus, TStE620, TStE690, Weldox 500, Hardox, L480 - L550, X65, X80, X90, X100, Hardox 400, XAR 400, Dilidur 400, Domex 600MC, Domex 650 MC, 20MnCr65, 28CrMn43, ASTM: A 517, A 537, A 625, HY100, (16NiCrMo12-6), Oceanfit 100, Oceanfit 690
APPROVALS	LRS (5Y69), ABS (5Y69), DNV, DB, TÜV, GL, BWB-WWEB, CE approved

WELDING POSITIONS:

WELD METAL ANALYSIS % (TYPICAL VALUES FOR M21)

C	Mn	Si	Cr	Ni	Mo	P	S
0.05	1.6	0.4	0,5	2,2	0,5	<0.015	<0.015

MECHANICAL PROPERTIES

Heat Treatment	$R_{P0,2}$ (N/mm²)	Rm (N/mm²)	A5 (%)	Impact Energy (J) ISO-V -20°C	-40°C	-60°C	Hardness HRc / HV
AW	>690	780-960	>17		>69	>69	
SR	>670	760-850	>17		>60	>47	

AW: as welded, SR: stress relieved 580ºC / 2hr

WELDING PARAMETERS / PACKING

Welding Parameters D (mm)	Voltage (V)	Current (A) DC+	Packing spool type	kg / spool /drum	kg / pallet
1,0	14-26	70-230	D-200 / K-300 / Drum	5 / 16 / 300	1000 / 1024 / 600
1,2	14-31	90-310	D-200 / K-300 / Drum	5 / 16 / 300	1000 / 1024 / 600
1.6	17-36	120-380	D-200 / K-300 / Drum	5 / 16 / 300	1000 / 1024 / 600

REDRYING TEMPERATURE	Not required
GAS ACC. EN ISO 14175:	M21

1

Figure D - 2: S690 Filler Material

Annex E – Basic Components EN1993 1-8 (2005)

Table 6.1: Basic joint components

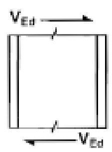
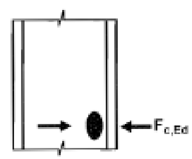
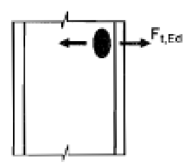
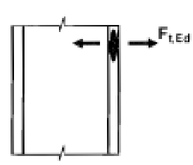
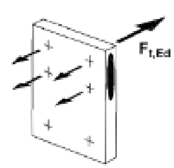
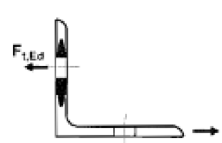
Component			Reference to application rules		
			Design Resistance	Stiffness coefficient	Rotation capacity
1	Column web panel in shear		6.2.6.1	6.3.2	6.4.2 and 6.4.3
2	Column web In transverse compression		6.2.6.2	6.3.2	6.4.2 and 6.4.3
3	Column web in transverse tension		6.2.6.3	6.3.2	6.4.2 and 6.4.3
4	Column flange in bending		6.2.6.4	6.3.2	6.4.2 and 6.4.3
5	End-plate in bending		6.2.6.5	6.3.2	6.4.2
6	Flange cleat in bending		6.2.6.6	6.3.2	6.4.2

Figure E - 1: Table 6.1 - Basic Components - Part 1

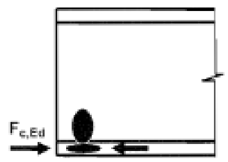
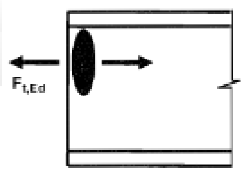
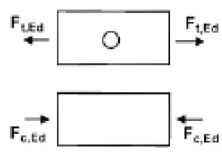
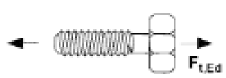
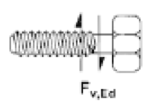
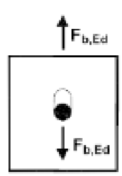
Component			Reference to application rules		
			Design Resistance	Stiffness coefficient	Rotation capacity
7	Beam or column flange and web in compression		6.2.6.7	6.3.2	*)
8	Beam web in tension		6.2.6.8	6.3.2	*)
9	Plate in tension or compression		in tension: - EN 1993-1-1 in compression: - EN 1993-1-1	6.3.2	*)
10	Bolts in tension		With column flange: - 6.2.6.4 with end-plate: - 6.2.6.5 with flange cleat: - 6.2.6.6	6.3.2	AC2) 6.4.2 (AC2
11	Bolts in shear		3.6	6.3.2	6.4.2
12	Bolts in bearing (on beam flange, column flange, end-plate or cleat)		3.6	6.3.2	*)
*)	No information available in this part.				

Figure E - 2: Table 6.1 - Basic Components - Part 2

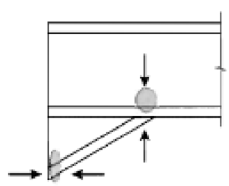
Component			Reference to application rules		
			Design Resistance	Stiffness coefficient	Rotation capacity
13	Concrete in compression including grout		6.2.6.9	6.3.2	*)
14	Base plate in bending under compression		6.2.6.10	6.3.2	*)
15	Base plate in bending under tension		6.2.6.11	6.3.2	*)
16	Anchor bolts in tension		6.2.6.12	6.3.2	*)
17	Anchor bolts in shear		6.2.2	*)	*)
18	Anchor bolts in bearing		6.2.2	*)	*)
19	Welds		4	6.3.2	*)
20	Haunched beam		6.2.6.7	6.3.2	*)
*)	No information available in this part.				

Figure E - 3: Table 6.1 - Basic Components - Part 3