الـجـمـهـورية الجـزائـريـــة الـديـمـقـراطـيــة الــشـعـبـيــة

People's Democratic Republic of Algeria

Ministry of Higher Education and Scientific Research

UNIVERSITY OF GHARDAIA

FACULTY OF SCIENCES AND TECHNOLOGY

DEPARTMENT OF HYDRAULIC AND CIVIL ENGINEERING

A THESIS

Submitted in Partial Fulfilment of the Requirement for the Degree of

MASTER

Field: **Civil Engineering**
Option: **Structures**

Presented by

Abdelaziz Mohammed Amine

And

Chikh Salah Nacereddine

Title

Finite Element Analysis of Semi-Rigid Steel Beam-to-Column Connections

The academic year 2021/2022

Mr Bahaz Abdesselam **Doctor (MAB), University of GHARDAIA** **Supervisor**

الـجـمـهورية الجـزائـريـــة الـديمـقراطيـة الــشعبيــة

République Algérienne Démocratique et Populaire

Ministère de l'Enseignement Supérieur et de la Recherche Scientifique

UNIVERSITÉ DE GHARDAIA

FACULTÉ DES SCIENCES ET DE LA TECHNOLOGIE

DEPARTMENT D'HYDRAULIQUE ET GENIE CIVIL

THÈSE

Présentée pour l'obtention du diplôme de

MASTER

Filière : **Génie civil**
Option : **Structures**

Présentée par

Abdelaziz Mohammed Amine

And

Chikh Salah Nacereddine

THÈME

Analyse Des Assemblages Poteau-Poutre Semi Rigides En Acier Par La Méthode Des Eléments Finis

Année Universitaire 2021/2022

Mr Bahaz Abdesselam	**Docteur (MAB), Université de GHARDAIA**	**Encadreur**

ACKNOWLEDGEMENTS

Start with the Name of Allah, the Most Beneficent, the Most Merciful.

We would like to thank, in particular, those who contributed to this thesis:

- Our supervisor, Dr Bahaz Abdesselam, offered us his knowledge and supported us during our work on the thesis and throughout our academic careers. His door has always been opened for us. The completion of this thesis has been a journey, and Dr Bahaz Abdesselam has been an ideal travelling companion.
- Members of the thesis's evaluation committee for their participation, for their valuable comments, and remarks.
- Department members, for sharing their many years of wisdom and providing the best education possible. They have all been an inspiration.

Finally, special thanks and loves go to our parents, brothers, sisters and other relatives for their sacrifice, patience, and constant inspiration during my most critical time, without their understanding and support. We might not have come to this position.

مـلخـــــــص

تم تصميم الهياكل الفولاذية بشكل تقليدي على افتراض أن الوصلات عمود-عارضة مثبتة بشكل مفصلي أو صلبة تمامًا. هذا يبسط التحليل وعمليات التصميم الإنشائي ولكن على حساب عدم الحصول على فهم مفصل لسلوك الوصلات ، والتي في الواقع لها صلابة محدودة وبالتالي فهي شبه صلبة. شهد القرن الماضي تطور طرق تحليل المفاصل شبه الصلبة. تتفق الدراسات على أنه ينبغي أخذ سلوك دوران الوصلات بعين الإعتبار في تحليل الهياكل المعدنية. يتم ذلك عادةً باستخدام منحنى عزم الدوران. يمكن استخدام نماذج تحليلية ,تجريبية ,معلوماتية, ميكانيكية أورقمية لتحديد السلوك الميكانيكي للوصلة. الأكثر شيوعًا هو النموذج الميكانيكي ، مع العديد من الاختلافات (مثل طريقة المركبات).

هذه الأطروحة تهدف إلى توصيف الوصلات الطرفية من النوع عمود-كمرة باستخدام طريقة المركبات المعتمدة في EC03.بعد ذلك ، تم تطوير نموذج العناصر المنتهية ثلاثية الأبعاد إعتمادا على النموذج الميكانيكي لطريقة المركبات باستخدام حزمة البرنامج متعدد الأغراض ABAQUS . يأخذ نموذج العناصر المحدودة المقترحة (FE) في الاعتبار المواد غير الخطية الهندسية ، والعيوب الأولية ، والتلامس بين الأسطح المجاورة وقوة الشد في البراغي. تم التحقق بنجاح من صحة النموذج المقترح و ذلك من خلال مقارنته مع النتائج التي تم الحصول عليها من طريقة المركبات. تم استخدام تنظيم EC3 الجزء 1-8 لتقييم مقاومة مفصل الصفيحة الطرفية والصلابة الأولية. تم أيضا فحص النتائج فيما يتعلق بأنماط الفشل ، وتطور المقاومة ، والصلابة الأولية ، وقدرة الدوران.

المفاتيح الدلالية

وصلة النهاية بصفيحة، طريقة المركبات، التحليل الرقمي، الوصلة النصف صلبة، برنامج ABAQUS.

RÉSUMÉ

Les portiques en acier étaient traditionnellement conçus en supposant que les assemblages poteau-poutre sont idéalement articulés ou entièrement rigides. Cela simplifie les processus d'analyse et de conception structurelle. Cependant, une compréhension globale du comportement des assemblages a été obscurcie, qui en réalité, ont une rigidité finie et sont donc semi-rigides. Le dernier siècle a vu l'évolution des méthodes d'analyse des assemblages semi-rigides. Les études conviennent que dans l'analyse de l'ossature, le comportement de rotation des assemblages doit être pris en compte. Cela se fait généralement en utilisant la courbe moment-rotation. Des modèles analytiques, empiriques, expérimentaux, informationnels, mécaniques et numériques peuvent être utilisés pour déterminer le comportement mécanique des assemblages. Le plus populaire est le modèle mécanique, avec plusieurs variantes (par exemple, la méthode des composants).

Cette thèse vise à caractériser les assemblages poteau-poutre avec platine d’extrémité en utilisant la méthode des composants de l'EC03. Ensuite, un modèle d'éléments finis 3D est développé sur la base du modèle mécanique de la méthode des composants à l'aide du logiciel polyvalent ABAQUS. Le modèle d'éléments finis proposé prend en compte les non-linéarités matérielles et géométriques, l'imperfection initiale, le contact entre les surfaces adjacentes et la force de précontrainte dans les boulons. Le modèle numérique proposé a été validé avec succès par rapport aux résultats obtenus à partir de la méthode des composants. L'approche de EC3 partie 1.8 est utilisée pour évaluer la résistance et la rigidité initiale de l'assemblage boulonné ave platine d’extrimité. Les résultats sont examinés en ce qui concerne les modes de rupture, l'évolution de la résistance, la rigidité initiale et la capacité de rotation.

Mots-clés

Assemblage par platine d’extrémité, méthode des composants, analyse numérique, assemblage semi-rigides, ABAQUS.

ABSTRACT

Steel portal frames were traditionally designed assuming that beam-to-column joints are ideally pinned or fully rigid. This simplifies the analysis and structural design processes but at the expense of not obtaining a detailed understanding of the behaviour of the joints, which in reality, have finite stiffness and are therefore semi-rigid. The last century saw the evolution of analysis methods of semi-rigid joints. Studies agree that in frame analysis, joint rotational behaviour should be considered. This is usually done by using the moment–rotation curve. Models such as analytical, empirical, experimental, informational, mechanical and numerical can be used to determine joint mechanical behaviour. The most popular is the mechanical model, with several variances (e.g. Component Method).

This thesis aims to characterize the end-plate beam-to-column joints using the components method of EC03. Then, a 3D finite element model is developed based on the mechanical model of the component method using the multi-purpose software package ABAQUS. The proposed finite element (FE) model takes into account material and geometrical non-linearities, initial imperfection, contact between adjacent surfaces and the pretension force in the bolts. The proposed FE model has been successfully validated against the results obtained from the component method. The EC3 part 1.8 approach is used to evaluate the end-plate bolted joint's resistance and initial stiffness. The results are examined with respect to the failure modes, the evolution of the resistance, the initial stiffness, and the rotation capacity.

Keywords

End-plate joint, component method, numerical analysis, semi-rigid joint, ABAQUS.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

LIST OF NOTATIONS

μ : Tangential friction coefficient of contact surfaces	36
E : Young modulus	20
EC : European code	20
EN: European normalization	12
FE : Finite element	37
F_{Rd} : Design resistance	25
f_{ub} : Ultimate strength of the bolt	17
f_{yb} : Yield strength of the bolt	17
h_1: Distance between the first horizontal row of bolts and the top edge of end-plate	34
h_b : Beam height	34
HR: High resistance	34
h_c : Column height	34
h_p : End-plate height	34
K : Stiffness	25
Mj,Rd : Design moment resistance of the joint	23
$M_{p,beam}$: Plastic moment of beam	21
P_i : Distance between horizontal rows of bolts	34
$S_{j,ini}$: Initial stiffness of the joint	20
t_{wb} : Thickness of the beam web	34
W: Distance between two vertical rows of bolts	34
φ : Joint rotation	21
I_{xb} : Beam area moment of inertia about x-axis	34
I_{xc} : Column area moment of inertia about x-axis	34
a_f : Flange weld neck	34
a_w : Web weld neck	34
b_b : Beam width	34
b_c : Column width	34
r_b : Root fillet radius of the beam	34
r_c : Root fillet radius of the column	34
t_{fb} : Thickness of the beam flange	34
t_{fc} : Column flange thickness	34
t_p : End-plate thickness	34
t_{wc} : Web column thickness	34

CHAPTER 1 INTRODUCTION

The steel structure is a metal structure made of structural steel components (Beams, columns, plates) that connect to carry loads and provide full rigidity and high strength grade of steel; this structure requires less raw materials than other materials types of structure like concrete structure.

Structural steel components are connected together at joints in a number of ways and by using a variety of connectors, pins, rivets, bolts, and welds of various types.

Steel structures are used for every type of structure (High-rise buildings, equipment support systems, infrastructure, bridges, towers, airport terminals, pipe racks, etc.)

Most countries have famous and memorable steel structures worldwide (Eiffel Tower Paris, Harbour Bridge Sydney, Brooklyn Bridge New York City, etc.).

Structural steel is the best choice for many reasons; this is why it is preferred over other types of structures; here are the top reasons:

- As a first importance safety
- Environment-friendly
- Steel is profoundly strong, more affordable to remodel
- Steel is value for money

It has been more than 100 years since the steel structure was discovered in this world. As early as 19 century, the high-rise building in Chicago applied a steel structure.

Since the 1960s, America began to design and construct high-rise steel structure with more than 80 floors.

Figure 1:1 Structural steel components

Nowadays, in actual projects, steel structures are covered within European Normalization (EN); there are twelve constitutive parts: The main sections of the EN document are:

■ EN 1993-1 Design of Steel Structures: General rules and rules for buildings

■ EN 1993-2 Design of Steel Structures: Steel bridges

■ EN 1993-3 Design of Steel Structures: Towers

■ EN 1993-4 Design of Steel Structures: Silos, tanks

■ EN 1993-5 Design of Steel Structures: Piling

■ EN 1993-6 Design of Steel Structures: Crane supporting structures

In this work, the focus in Part 1-1 and Part 1-8 General rules and rules for buildings and design of joints.

■ EN 1993-1-1 Design of Steel Structures: General rules and rules for buildings

■ EN 1993-1-2 Design of Steel Structures: Structural fire design

■ EN 1993-1-3 Design of Steel Structures: Cold-formed thin gauge members and sheeting

■ EN 1993-1-4 Design of Steel Structures: Stainless steel

■ EN 1993-1-5 Design of Steel Structures: Plated structural elements

■ EN 1993-1-6 Design of Steel Structures: Strength and stability of shell structures

■ EN 1993-1-7 Design of Steel Structures: Strength and stability of planar plated structures transversely loaded

■ EN 1993-1-8 Design of Steel Structures: Design of joints

■ EN 1993-1-9 Design of Steel Structures: Fatigue strength of steel structures

■ EN 1993-1-10 Design of Steel Structures: Selection of steel for fracture toughness

■ EN 1993-1-11 Design of Steel Structures: Design of structures with tension components made of steel

■ EN 1993-1-12 Design of Steel Structures: Supplementary rules for high-strength steel.

This chapter is interested in general types of steel assembly in Structural steel.

Firstly, giving a definition of the functioning and significance of components in this type of construction, and then we present different joint methods.

Finally, the focus is on the most important connections in a steel construction that resist loads (normal stress and shear stress).

1.1 Definition of the Role of Steel Joint Components

The Characteristics of steel structures are composed of elements (beam-column) made of rolled or welded profiles and different forms (I or H) that must be connected together to form the steel frame.

The connections between these different represented elements are currently called steel joints. These are very important components in steel construction. A poorly designed, realized, or calculated assembly can lead to the collapse of the structure. As a result, the design and calculation of steel joints are capital important.

1.2 Different Connections Forms Encountered in Steel Construction

In steel structures building, the structural elements are connected by various forms of steel connections. Depending on the nature of the assembled components, as shown in Figure 1:2

- Beam-to-column connection (A)
- Beam-to-beam or (beam splice) connection (B)
- Column splice connection (C)
- Column base plate connection (D)

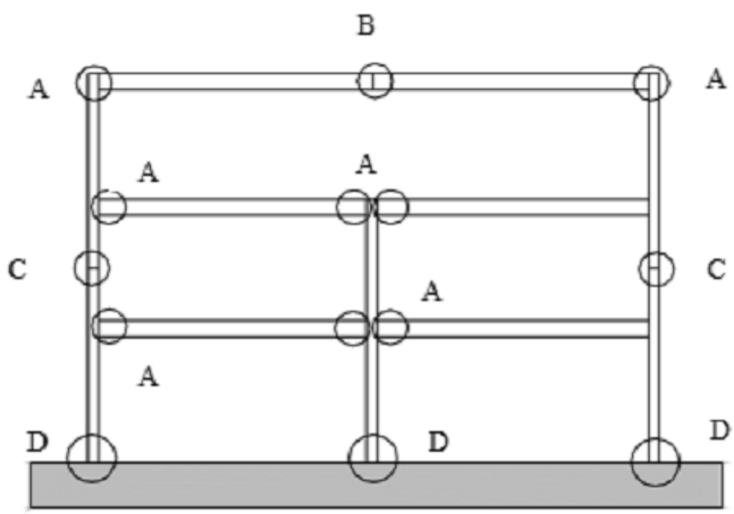


Figure 1:2 Different types of connections in a metal frame

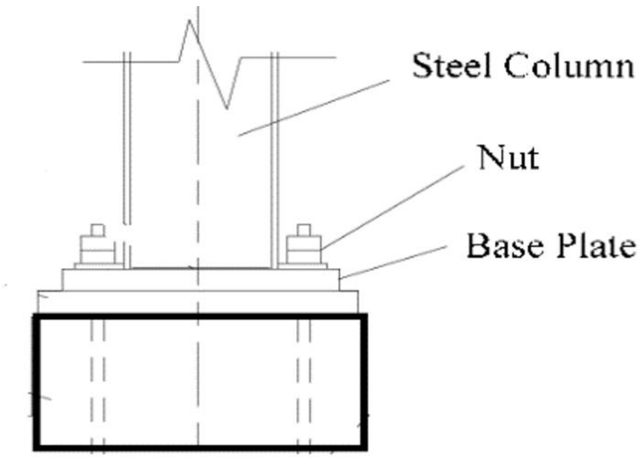
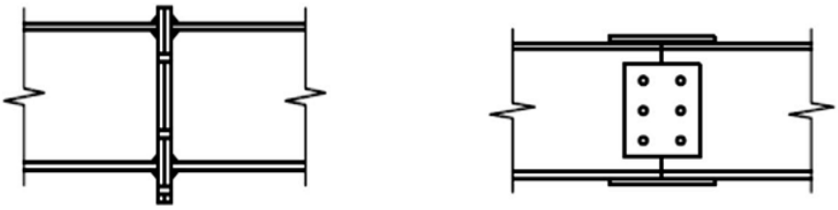


Figure 1:3 Column base connection



a) End –plate type beam splice

b) Cover –joint type beam splice

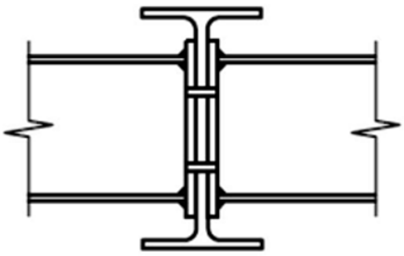


Figure 1:4 Examples of steel beam–to-beam joints

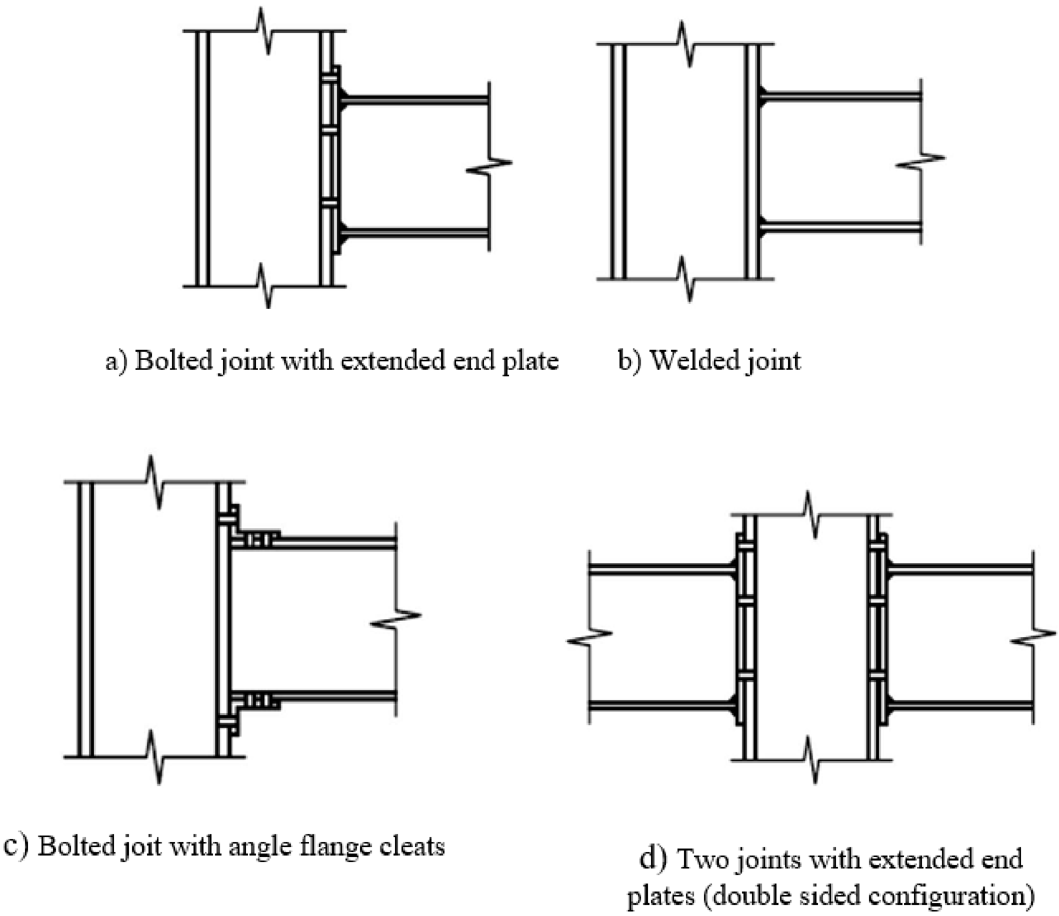
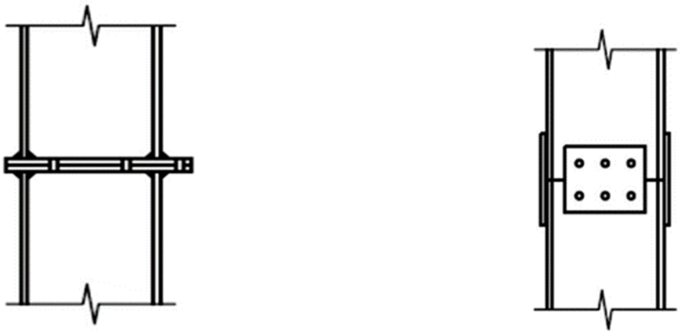


Figure 1:5 Examples of steel beam-to-column joints



a) End –plate type column splice b) Cover –joint type column splice

Figure 1:6 Examples of steel column–to–column joints

1.3 **Types of Connections**

Connections are the most important and basic steel structure or element needs. Without connection, the steel structure cannot construct properly according to the design methods given in chapter 3 [1]:

1.3.1 Riveted connection

Riveting is the particular method of connecting together pieces of metal. This process is conducted by inserting the ductile steel pins called rivets into the holes of pieces to be joined and forming the head at the end of the rivet to prevent each metal piece from coming out. They are used usually in old structures. Their diameter generally varies from 10 to 28 mm.

1.3.2 Bolted connection

Bolts are fasteners with external threads designed for driving through holes in assembled parts. Bolts classes and nominal values of yield strength f_{yb} and ultimate tensile strength f_{ub} are included in Table 1:1. The first number is the ultimate tensile strength f_{ub} divided by 100. The fractional part indicates the ratio f_{yb}/f_{ub}; for example, for a 10.9 bolt, the ultimate tensile strength f_{ub} is 10×100 = 1000 (N/mm2), and the yield strength f_{yb} is obtained as 1000×0.9 = 900 (N/mm2).

We distinguish two types of bolts according to their mechanical properties:

- Ordinary bolt refers to the bolt with low strength level requirements and generally with (4.6 and 6.8) class.
- High strength bolt (HS Bolts) High strength bolts are made from carbon steel or tempered alloy steel., And in general between (8.8-10.9) class.

Table 1:1: Properties for bolt steel according to EC03 part 1-8

Bolt class	**4.6**	**4.8**	**5.6**	**6.5**	**6.8**	**8.8**	**10.9**
f_{yb} (N/mm²)	240	320	300	300	480	640	900
f_{ub} (N/mm²)	400	400	500	600	600	800	1000

EN 1993-1-8 [1] distinguishes different categories of bolted connections. The two main categories. The loads applied to the bolt are shear and tension connections.

Generally, ordinary bolts used for the high strength bolts are used for structural dynamics Also bridge assemblies.

1.3.3 **Welded connection**

Welding is a process that enables connecting metallic components of the joint, usually through heat. In some welding processes, a filler material is added to facilitate joining. The assemblage of parts that are joined by welding is called a Weldment. Welding requires a source of sufficient heat to fuse the material that may be of electrical origin.

Welding implies:

- The existence of a sufficient heat source to obtain the fusion of the material can be of different origins.
- Electrical origin: (Arc, Plasma or Resistance Spot Welding)
- Gas welding: (M. A.G) Metal Active Gas welding method
- Mechanical origin: Friction, welding;

We called solderability material an aptitude ability of the material to be welded. Welding is useful more than bolts cause has many advantages:

- Welded joints are rigid.
- The welded joint has high strength, sometimes more than the parent metal.
- Welds usually have an excellent aesthetic appearance than bolts.

However, the welded connection has the following disadvantages:

- The possibility of brittle fracture is more in the case of welded connections.
- The base of elements must be solderability.
- Highly skilled persons are required for welding is also expensive.

1.4 Conclusion

In steel construction, the design of connections should play a prominent part in the safe and efficient assembly of structural building components; the following cases should be controlled:

- Make sure that all joining methods are in a good way.
- The joint must carry the moments, forces and shears arising from the frame analysis.
- Make sure of an excellent aesthetic of the construction.

CHAPTER 2 COMPONENT METHOD

2.1 Introduction

The steel joints must be modelled for the overall framework analysis. Joints modelling can be divided into simple, semi-continuous, and continuous. The type of joint modelling adopted depends on both the type of frame analysis and the assembly class in terms of rigidity or resistance, as described below.

2.2 Classification of Steel Joints

2.2.1 Classification by stiffness

EC3 offers a practical classification of assemblies in terms of their initial stiffness. It makes it possible to classify steel connections into three categories according to their (stiffness): Articulated (pinned), rigid or semi-rigid. The hinge support is able to resist. It is also used in doors to produce only rotation in a door. Hinge support reduces sensitivity to the earthquake. Joints are classified according to EC3 [1].

Table 2:1 Steel joints classification according to EC3 [1].

Rigid joint	$S_{j,ini} \geq 25\ EI/L$ $S_{j,ini} \geq 8\ EI/L$	(unbraced frames) (braced frames)
Semi-rigid joint	$0.5\ EI/L < S_{j,ini} < 25\ EI/L$ $0.5\ EI/L < S_{j,ini} < 8\ EI/L$	(unbraced frames) (braced frames)
Pinned joint	$S_{j,ini} \leq 0.5\ EI/L$	

2.2.1.1 Pinned joint

A joint can be considered articulated if it cannot develop significant moments that would be likely to exert an undesirable influence on the elements of the structure.

Articulated assemblies can transmit only the calculated forces, normal and sharp forces, and accept the consequent rotations.

2.2.1.2 Rigid assemblies

They are called rigid assemblies if the deformation does not influence the distribution of forces and moments in the structure or the overall deformation of the structure. Deformations of rigid assemblies should not reduce the structural strength by more than 5%.

2.2.1.3 Semi-rigid assemblies

Assemblies that do not meet the criteria for rigid assemblies All pinned connections will be classified as semi-rigid assemblies. Semi-rigid assemblies must be able to predict the level of interaction between structural elements based on the moment-rotation characteristics of the nodes. They are able to transmit only the efforts and moments calculated during their conception.

2.2.1 **Classification by strength**

For classifying connections according to strength, it is common to non dimensionalize the vertical axis of the $M - \varphi$ curve by the beam plastic moment capacity, $M_{p,\,beam}$, as is shown in Figure 2:1. Connections not capable of transmitting at least 25% of the design resistance for full strength connections are classified as nominally pinned. A nominally pinned joint shall be capable of accepting the resulting rotations under the design loads. A joint which does not meet the criteria for a full-strength joint or a nominally pinned joint should be classified as a partial-strength joint.

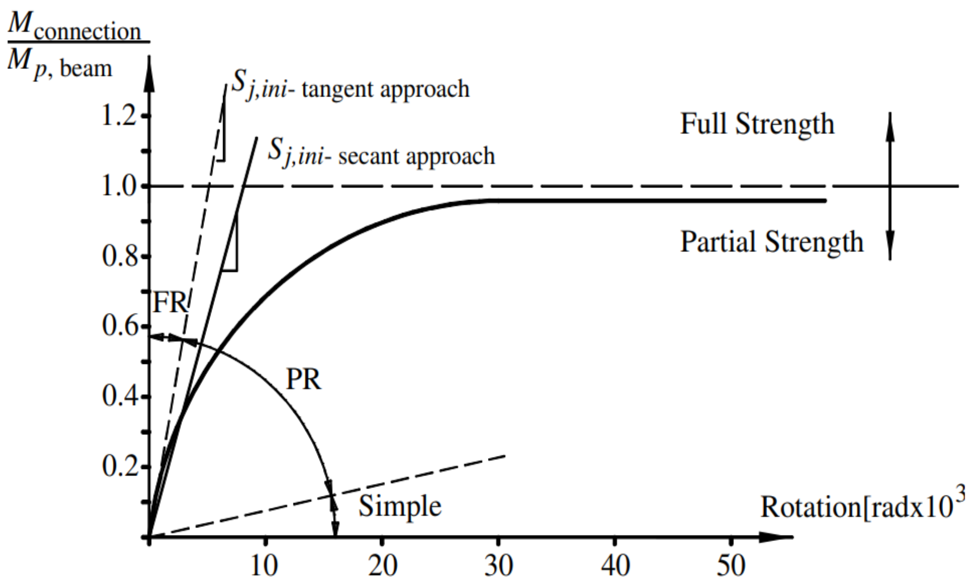


Figure 2:1 Connection classification by stiffness and strength [2]

2.3 Components Method

The component method was originally developed for the calculation of steel and composite joints, the characterization of the response of the joints in terms of stiffness, resistance and ductility. In the classical component method, the characteristic behaviour of a complex joint is assembled from a number of single components, of which mechanical and geometrical properties are known. The characteristic behaviour of the various components can be determined either by *experimental*, *numerical* or *analytical methods.* Using the component method for the design of a moment resisting joint, the strength and stiffness, either in tension, compression or shear, have to be calculated for every basic component. These calculation results determine the properties of the elastic or plastic springs used to assemble the spring model for the complete joint

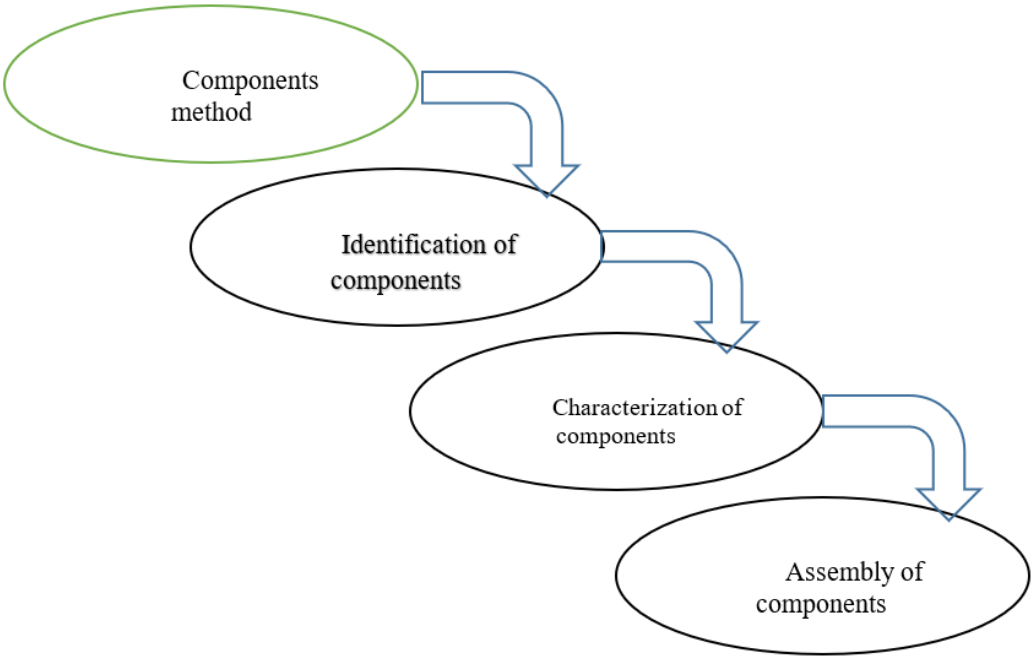


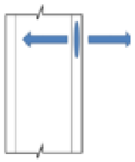
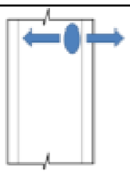
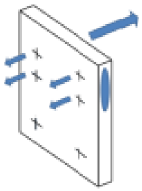
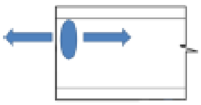
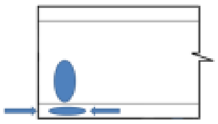
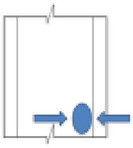
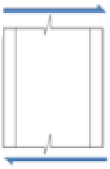
Figure 2:2 Steps of the component method

- *Identification of components:* Defined different types of forces in every component in the structure (compression traction and shear)
- *Properties of the components:* Defined all the mechanical properties of components.
- *Assembly of the components:* After the mechanical properties of all components have been determined, the various components can be assembled to determine the moment resistance ($M_{j,\,Rd}$), initial rotational stiffness of joints ($S_{j,ini}$) and deformation capacity of the whole connection (φ).

2.3.1 **Classification of components**

Deferent types of mechanical properties of these components in steel structural construction rules for calculating each component's strength, stiffness, and deformation capacity are given in EC3 [1]. The components method can identify components under moment strength (resistance) $M_{j,Rd}$, the deformation; rotational stiffness $S_{j,ini}$, each of these components are modelled using every component to calculate each component more easily.

Table 2:2 Basic components of the studied joint and their design resistances [3]

TENSION COMPONENTS	
1. Bolts in tension $F_{t,Ed}$ 	$F_{t,Rd}$: Bolt tension resistance
2. Colum flange in bending  $F_{t,Ed}$	$F_{t,fc,Rd}$: Column flange tension resistance in bending
3. Column web in transverse tension  $F_{t,Ed}$	$F_{t,wc,Rd}$: Column web tension resistance
4. End-plate in bending $F_{t,Ed}$ 	$F_{t,ep,Rd}$: End plate tension resistance in bending
5. Beam web in tension $F_{t,Ed}$ 	$F_{t,wb,Rd}$: Beam web tension resistance
COMPRESSION COMPONENTS	
6. Beam flange and web in compression $F_{c,Ed}$ 	$F_{c,fb,Rd}$: Beam flange compression resistance
7. Column web in transverse compression $F_{c,Ed}$ 	$F_{c,wc,Rd}$: Column web compression resistance
SHEAR COMPONENTS	
8. Column web panel in shear V_{Ed}  V_{Ed}	$V_{wp,Rd}$: Column web panel shear resistance

2.3.2 Description of components method

The component method is a mechanical-analytical method that permits the determination of the mechanical behaviour of the connection. This method consists of the representation of connections in springs; each spring has its own resistance ($F_{Rd,i}$) and stiffness (K) to tension, compression, or shear stress.

The total resistance of the connection is obtained from the resistances of its components. The total resistance is conditioned to the resistance of the weakest link, similarly to the behaviour of chain links.

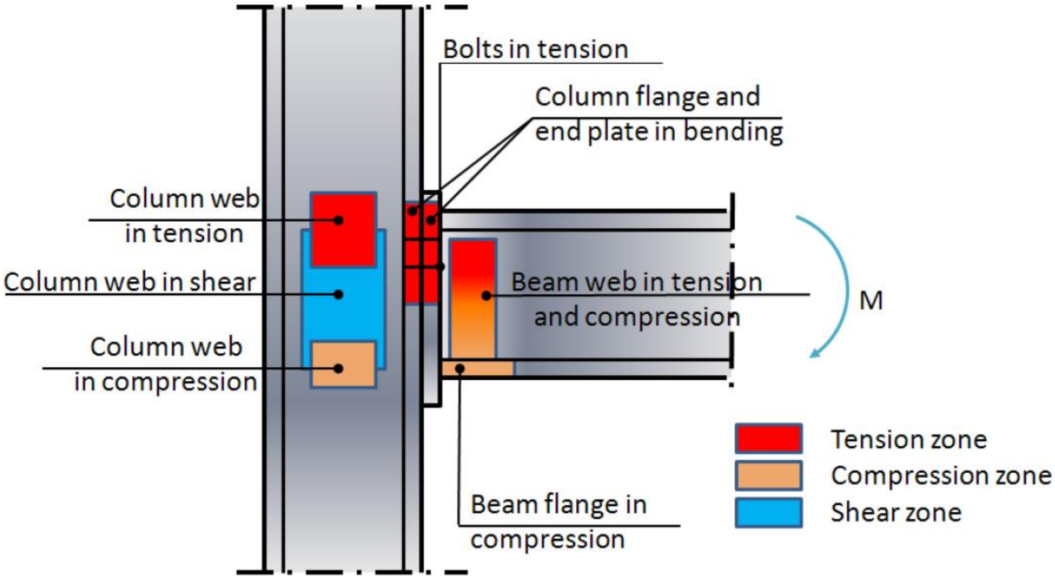


Figure 2:3 Basic components of a beam-to-column joint in bending [3]

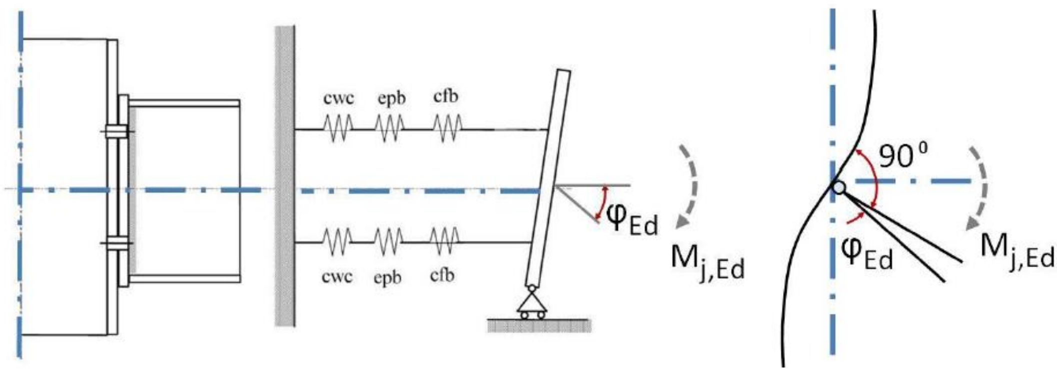


Figure 2:4 Joint configuration as a rotational stiffness string. The joint properties can be presented as a moment-rotation characteristic [3]

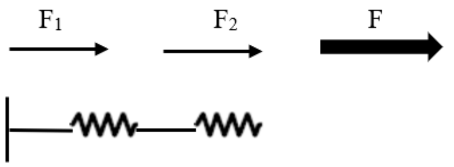
2.3.3 Assembly of components

The transition from the force-displacement relationships of individual components to the curve (moment-rotation) of the connection is obtained by satisfying the requirements of compatibility and equilibrium and the limitations of resistance and deformation capacity.

Assembly of the components is based on the global analysis of the structure, which relates to the knowledge of the behaviour curves (moment-rotation) determined by the following three approaches: (1) the exploitation of the results obtained from experimental tests, (2) a numerical approach by the application of the finite element method, and (3) an analytical approach through the mechanical models with springs [1].

This mechanical model uses a spring at the level of each row of bolts; the latter consists of a series spring or a parallel spring; this makes it possible to introduce nonlinear behaviour laws of the components and to evaluate the rotation capacity of the assembly in relation to the ruin of any one of these components.

In series:



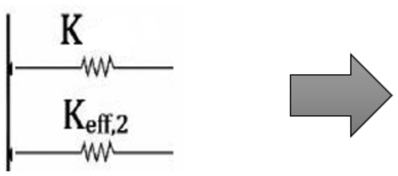
We have: $F = w.k.E$

And $w = w_1 = w_2$

$$w_{eq} = w = w_1 + w_2 \Rightarrow \frac{F_{eq}}{K_{eq}.E} = \frac{F_1}{K_1.E} + \frac{F_2}{K_2.E} \qquad \text{as} \quad F_{eq} = F_1 + F_2$$

$$\frac{F}{K_{eq}} = F\left[\frac{1}{K_1} + \frac{1}{K_2}\right] = \frac{1}{K_{eq}} = \frac{1}{K_1} + \frac{1}{K_2}$$

In parallel:



We have: $F = F_1 + F_2$

And: $w = w_1 = w_2$

$$F_{eq} = w_{eq} K_{eq} E = w_1 K_1 E + w_2 K_2 E \quad ; \text{ as } \quad w_{eq} = w = w_1 + w_2$$

$$\Rightarrow w_{eq} K_{eq} = w_1 K_1 + w_2 K_2 \Rightarrow K_{eq} = K_1 + K_2$$

In the last, evaluation of the key characteristics of the connection by finding $M_{j.Rd}$, and $S_{j.ini}$ with the procedure of assembly resistance $M_{j.Rd}$, which is associated with the design resistance of the connection component having the weakest resistance and the procedure of the initial stiffness $S_{j.ini}$ determined from the stiffness in the translation of the components of resistor assembly.

Table 2:3 Grouping in series and parallel of the joint components

	Groups	
	In parallel	**In series**
Initial rigidity k_{eff}	$K_1 + K_2$	$\frac{1}{\frac{1}{k_1}+\frac{1}{k_2}}$
Strength F_{eff}	F_1+F_2	F_1+F_2
Deformation capacity	$w_1 = w_2$	$w_1 + w_2$

CHAPTER 3 DEVELOPMENT OF THE FE MODEL

3.1 Introduction

The Characteristics of steel structures are composed of elements (beam-column) made of rolled or welded profiles and different forms (I or H) that must be connected together to form the steel frame. Finite element analysis (FEA) is the use of calculations, models and simulations to predict and understand how an object might behave under various physical conditions. Engineers use FEA to find vulnerabilities in their design prototypes.

FEA uses the finite element method (FEM). This numerical technique cuts the structure of an object into several pieces or elements and then reconnects the elements at points called nodes. The FEM creates a set of algebraic equations that engineers, developers and other designers can use to perform finite element analysis.

The physical experiences of a product, such as its structural or fluid behaviour and thermal transport, are frequently described using partial differential equations (PDEs). Finite element analysis emerged as a way for computers to solve both linear and nonlinear PDEs. However, it is important to note that FEA only provides an approximate solution; it is a numerical approach to finding the real results of partial differential equations.

Using finite element analysis can reduce the number of physical prototypes created and experiments performed while optimizing all components during the design phase. Finite element analysis software emerged in the 1970s with software such as ABAQUS, ADINA and ANSYS. Now, it is common to find virtual testing and design optimization integrated into the product development cycle to improve product quality and reduce the time it takes to enter the market.

3.2 FEA Principles

Finite element analysis is based on principles that include boundary conditions, such as forces and pressures, as well as three governing equations:

1. Equilibrium equations, which find when the opposing forces or influences are balanced.
2. Strain-displacement relations, which measure the deformation that the design experiences under any given external impact.
3. Constitutive equations, which are relations between two physical quantities, specific to the given metal or substance, which predict the material's response to external stimuli.

3.3 FEA Process

For finite element analysis to perform its necessary simulations, a mesh -- containing millions of small elements that together form the shape of a structure -- must be created. Calculations must be performed on every single element; the combination of each of these individual answers provides the final result for the full structure.

This process can be further divided into three steps: the preprocess, process and post-process.

During the preprocessing step, the user is asked to select the analysis type -- such as modal analysis or static structural analysis -- as well as the element type. Next, the material properties must be defined, and nodes must be made. The elements are then built by assigning connectivity to the nodes. Finally, boundary conditions and loads are applied.

The computer performs the second step, the process. The computer solves the boundary value problem and then presents the results to the user during this step.

During the postprocessing step, the user reviews the generated results and notes factors such as displacement, temperature, time history, stress, strain and natural frequency.

Designers using finite element analysis should be aware of inherent errors that can be found in this process, such as the simplification of geometry in the finite element method and use of basic integration techniques; errors in computing stemming from numerical difficulties or the limited number of digits available in computers; and common user mistakes, such as selecting the wrong type of element or providing inconsistent units of measurement.

3.4 Benefits of FEA

- Finite element analysis provides the safe simulation of potentially dangerous or destructive load conditions and failure modes, allowing engineers to discover a system's physical response at any location. Other benefits include:
- Increased accuracy due to the analysis of any physical stress that might affect the design.
- Improved design because developers can observe how stresses within one element will affect the materials in another connected element.
- Earlier testing in the development process. Virtual prototyping allows the designer to model various designs and materials in a matter of hours rather than the days or weeks it takes to produce hard prototypes.
- Increased productivity and revenue because FEA software allows developers to produce higher quality products in a shorter design cycle while also using less material.
- Enhanced insight into critical design parameters as the result of being able to model both the design's interior and the exterior. This allows designers to determine how critical factors affect the structure as a whole, and where failures might occur.
- Optimized use of models because one common model can be used to test several failure modes or physical events

- Fast calculation times and relatively low investment costs.
- Access to existing experimental results, which can be pulled from the parametric analyses of already validated models and applied to the new model.

3.5 Common Applications of FEA

FEA is commonly used in mechanical, aerospace, automotive, civil engineering projects, and biomechanics. Specifically, it is important to design machines, analyze fatigue for machines and their parts, certify load capacities for lifting cranes, build airport bridges, and determine brake or rotor lifetime. Other uses of finite element analysis include:

- Improving product safety
- Testing possible real-world conditions on the design
- Reducing design and manufacturing costs
- Evaluating and optimizing alternative designs and materials
- Analyzing different basic solutions quickly
- Gaining product and company recognition and credit from locally and globally recognized certification authorities
- Optimizing operations on workstations and personal computers.

3.6 Introduction to Abaqus

The future is digital tools and virtual reality; simulation and numerical analysis have been developed in recent years in private or public industrial research (car manufacturers, aeronautics, space, etc.), thus improving industrial productivity and the lives of all consumers.

The finite element analysis solution optimizes virtual prototyping and numerical simulation for the enterprise using the assumptions of fracture mechanics in elasticity and dynamics. Thus, it stimulates and improves performance.

Simulation and digital analysis have a direct impact on the quality of components, materials and products subject to high operational requirements (safety, damage, etc.). and time to market of products; moreover, they increase the performance of product lifecycle management solutions. They appeared in the desire to minimize the cost of a study with an optimized number of trials. The difficulty is creating a CAD model coming as close as possible to reality (materials, boundary conditions, etc.). The study will only be validated on the basis of physical tests resulting in measurements for the Comparison between reality and simulation (crash test, etc.).

From a technical point of view, the results obtained thanks to ABAQUS are the balance of energies, nodal forces, deformations, displacements, stresses, speeds, accelerations and all the physical quantities necessary for the design of a model.

ABAQUS is a computer code using the finite element method created in 1978. It is software for simulating very varied problems in mechanics. Simulate the physical response of structures subjected to loads, temperatures, impacts or other external conditions. It is known and widespread, in particular, for its efficient treatment of nonlinear problems

ABAQUS is, therefore, a finite element calculation software package which consists of three products: ABAQUS/Standard, ABAQUS/Explicit and ABAQUS/CAE.

ABAQUS/Standard is a general-purpose solver that uses a traditional implicit integration scheme.

ABAQUS/Explicit is a solver that uses an explicit integration scheme to solve nonlinear dynamic or quasi-static problems.

ABAQUS/CAE constitutes an integrated visualization and modelling interface for the said solvers.

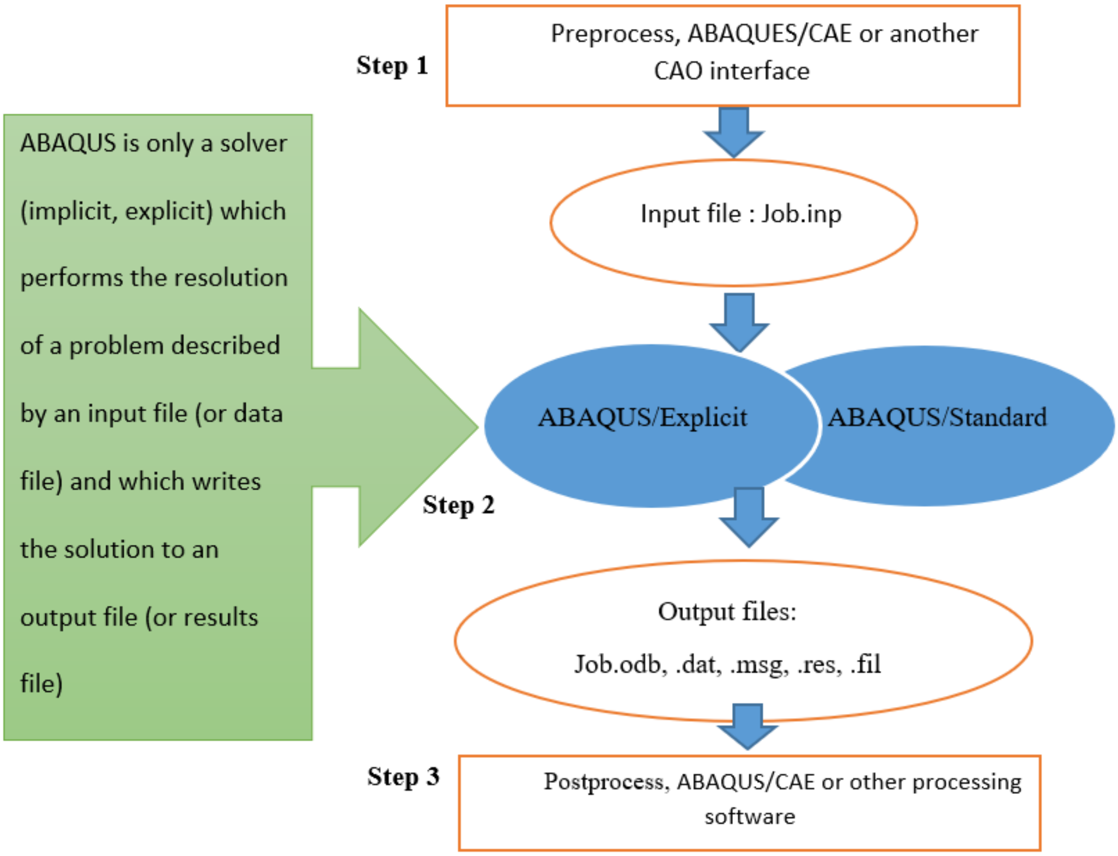


Figure 3:1 FE simulation procedure using ABAQUS [4]

This product is supplemented by additional and/or optional modules specific to certain applications.

3.7 Development of The Numerical Model

In order to understand and know the behaviour of bolted joints in different loads, the Finite Element Method software ABAQUS was used to model a column-beam joint based on the analytical model of A. Bahaz [5]. The initial data for this model is given in Table 3:1.

3.7.1 Material properties:

The standard mechanical properties for the joint which were obtained from the analytical model of A. Bahaz [5] are given in Table 3:1, Table 3:2 and Figure 3:2 Configuration of the studied end-plate joint . The plasticity behaviour in the joint was represented by the isotropic work hardening assumption and the von Mises yield criterion.

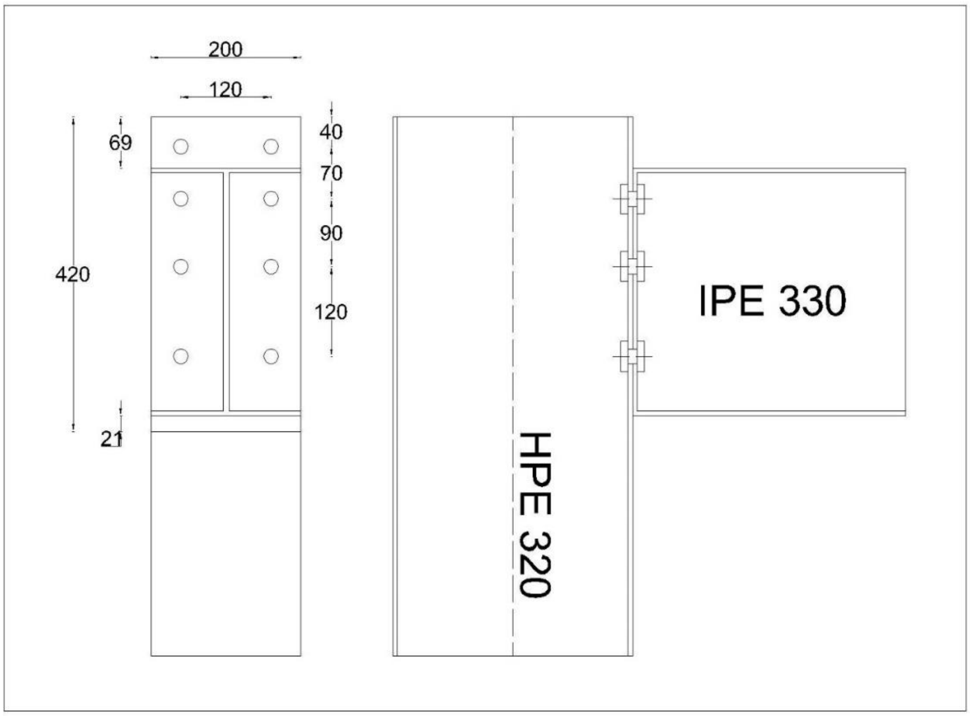


Figure 3:2 Configuration of the studied end-plate joint [5]

Table 3:1 The initial data of the joint model beam-to-column

Components		f_y **(N/mm²)**	f_u **(N/mm²)**	**Length (m)**
Beam **IPE 330**	$h_b = 330mm$ $b_b = 160mm$ $t_{wb} = 8mm$ $t_{fb} = 12mm$ $r_b = 18mm$ $I_{xb} = 11766.9\ cm^2$	235	460	3.00
Column **HEB 320**	$h_c = 320mm$ $b_c = 300\ mm$ $t_{wc} = 12\ mm$ $t_{fc} = 21\ mm$ $I_{xc} = 30823.5\ cm^2$ $r_c = 27\ mm$	235	460	1.50
End plate	t_p= 20 mm h_p= 420 mm a_f= 8 mm a_w = 8 mm	235	460	/
Bolts **M16/HR 8.8**	Number of rows = 04 Number of columns = 02 Type HR8.8 Diameter M16 (ϕ =8mm) W= 120 mm P_i = 70, 90,120 mm h_1 = 40 mm	640	800	/

Table 3:2 : Material properties used for the joint

Materials	Elastic behaviour		Plastic behaviour	
	Young's modulus (MPa)	Poisson's Ratio	Yield Stress (MPa)	Plastic Strain
S235 steel	210000	0.3	235	0.000
			460	0.218
Bolt steel M16-8.8	210000	0.3	640	0.000
			800	0.377

The steel for all constructive elements was defined as steel Grade S235. The effective material properties of the steel joint parts are summarized in Table 3:2. The material test data is in the form of engineering stress (σ_{eng}) and strain (ε_{eng}). In the FE analysis, the material property must be defined using the true stress and the true plastic strain relationships [4]. The values of the true stress (σ_{true}) and the true plastic strain ($\varepsilon_{pl,\ true}$) are determined from the engineering stress and logarithmic strain relationship using the equation (3:1):

$$\sigma_{true} = \sigma_{end}(1+\varepsilon_{eng}) \tag{3:1}$$

$$\varepsilon_{pl,true} = \ln(1+\varepsilon_{eng}) - \frac{\sigma_{true}}{E} \tag{3:2}$$

The von Mises yield criterion with the isotropic hardening rule was also used for the reinforcing steel. An elastic-linear-work-hardening material was considered, with tangent modulus being equal to 1/10000 of the elastic modulus, in order to avoid numerical problems.

3.7.2 Contact and constants

After introducing the different geometries (column, beam, end-plate and bolts) and the mechanical characteristics of the components of the assembly in ABAQUS, the assembly of the parts between them can begin. Finite element modelling requires defining several quantities and parameters and understanding how different parts interact with each other, which is known as the mechanical contact effect.

Many contacts exist in the model since the connection is a bolted joint. Numerical modelling of any joint requires a realistic representation of the contactinteraction between the various components in order to allow the load to be transmitted from one part of the model to another.

The contact established was defined between bolts and end-plates, bolts and columns, column and end-plates, and beam and end-plate. The interaction at the contact parts of the model was defined as surface-to-surface contact with finite sliding. The pretension of high strength bolts and the friction between the connection components are the essential parameters which define the joint behaviour. The pretension force applied to the bolt creates a hard contact between the connected elements. The transfer of the forces is realized through friction due to the clamping action between the connection elements. Small sliding surface to surface contact was applied to all the surfaces which have small relative sliding. The tangential contact between the end-plate and the column flange was considered as frictional contact ($\mu = 0{,}2$), using penalty stiffness formulation. Hard contact using augmented Lagrange formulation was considered for the normal contact between the same components. The bolt head/nut was tied constrained to the end-plate / column flange, and the beam was tied to the end-plate. The tangential contact between the bolt hole and the bolt shank was considered frictionless. The *hard constraint* was considered between the rigid plate and the top column section.

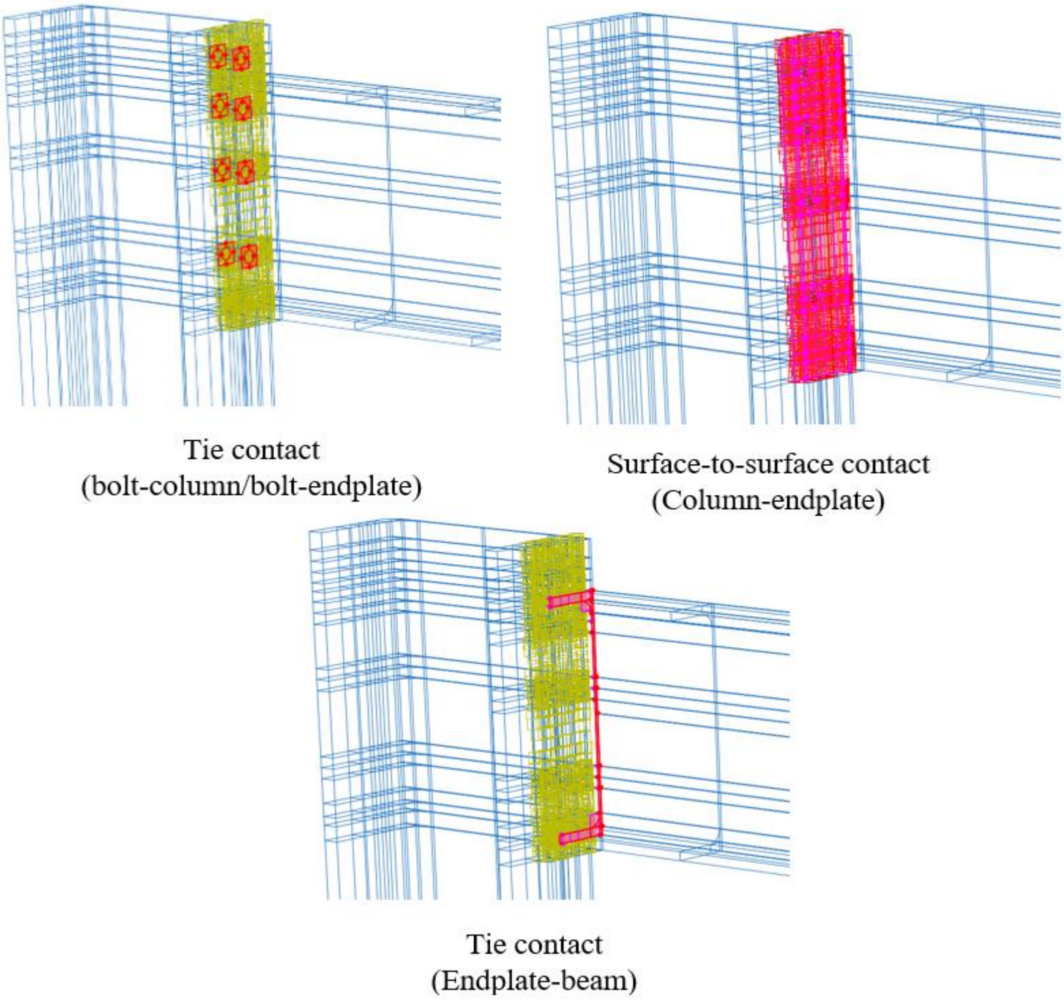


Figure 3:3 Interactions and constraints used to define all contacted surfaces.

3.7.3 **Boundary conditions and Load application**

A half part of the specimen was modelled using symmetry boundary conditions for Y-axis in light of geometrical symmetry. In order to prevent undesirable movements, the column had to be fixed at both top and bottom extremes during the whole simulation. This configuration was also established for the positioning conditions in the analytical model [5]. The column base was fixed using displacement and rotation restriction in every direction, while the upper part was restrained using displacement restriction in every direction.

The boundary conditions are crucial and significantly influence the modelling of the joint's behaviour. To achieve sensible behaviour in the joint region and to move away from any singularity problems, each bolt is restrained during the steps where the contact is established and then freed for later steps. The bolts restraint is defined at the nuts outer surfaces for displacement in every direction. Besides, to be more conservative during the contact process, the beams were also restrained at their extremes during these first contact steps and freed afterwards. Shows these boundary conditions, which concern the contact interaction process.

As can be noticed from the explanation above, the FE model boundary conditions vary during the complete analysis. At the beginning of the analysis, everything is fixed. This means column at both top and bottom sides, beam extremes and bolts. Two new steps are defined to establish the ultimate boundary conditions when the last bolt contact is established. In the first step, beams are released and then bolts.

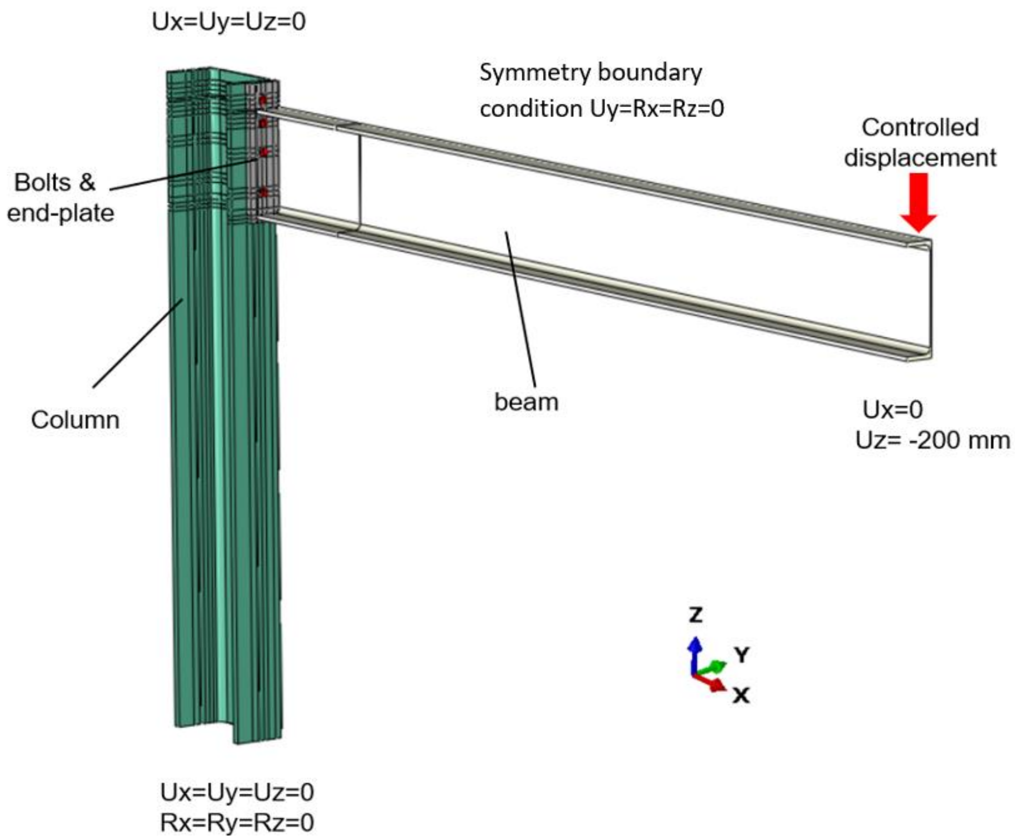


Figure 3:4 Detail of the FE model and boundary conditions

This step must define imposed conditions. These define the conditions imposed on the supports, the degrees of freedom (DOF), and how they are applied to the models. The boundary conditions are introduced by observing that there are three reference points. These reference points are the two ends of the column, and the third is that of the beam's free end (opposite of the joint). These points are located at the centre of the normal sections of the assembly elements. The final boundary conditions maintained are the column restrictions. Only then the loading can be applied.

The bolts are assigned the common bilinear stress-strain curve and modelled with C3D8R elements with the same bolt cross-section, head and nut. Both the head and the nuts are directly connected to the shank. The bolts are designed to have enough resistance and avoid failure (Mode 3 in EC03 [1]). A tightening moment of *200 N.m* is used to define the preloading force of the bolts. This value is calculated following the recommendations of EC3 [6]. In ABAQUS, bolt pretension can be defined as a *bolt load* (see Figure 3:5).

Imitating the analytical model of Bahaz et al. [5], the concentrated load was applied at the extreme of the beam at a distance of 3020 mm from the face of the column flange. The objective of these conditions is to analyze the behaviour of the joint up to the failure.

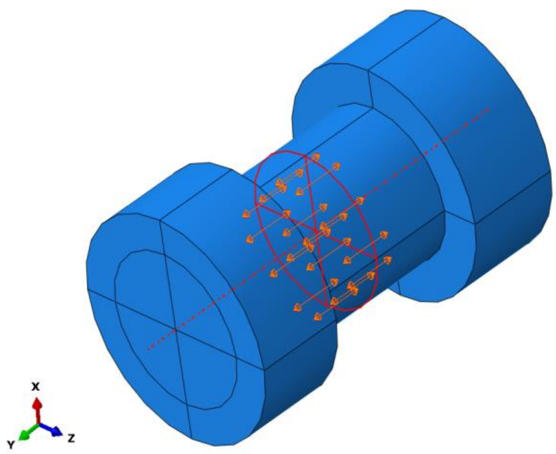


Figure 3:5 preloading cross-section

3.7.4 **Mesh**

The next step in creating the assembly components is the partitioning and meshing of each part. Meshing is the process by which the finite element modelling program divides parts of the model into finite elements. Computer calculation is able to take each piece of the assembly and analyze its shape. The calculation is based on the criteria used for the choice of the mesh of the finite elements for each part according to the nature of the problem treated. However, some shapes are too complex to mesh without assistance.

Therefore, partitioning is used to segment the part into geometric shapes such as rectangles which are easier to mesh. Segmentation is mainly used to facilitate the meshing process. During analysis, the part is always considered as a set of elements, and in addition, to facilitate the meshing process, partitioning can be used to separate surfaces that have different sizes from finite elements. In general, smaller finite elements are more accurate and better able to capture the stresses and strains that occur in the actual part of the experiment.

One of ABAQUS' powerful tools is the ability to choose the desired element type from its element library. ABAQUS offers several types of elements, such as solid elements, shell elements, and beam elements. In our study, all models are created using 3-D solid elements. Reduced integration is a type of numerical integration. This integral is defined as the sum of certain values calculated at specific points called Gaussian points. As the number of Gaussian points increases, the accuracy of the results will improve. There are two types of elements with reduced integration:

- Element with integration reduced to a single point of integration
- Element with total integration, which has two points of integration

The results are calculated at the point(s) of the element and then extrapolated to other points using shape functions.

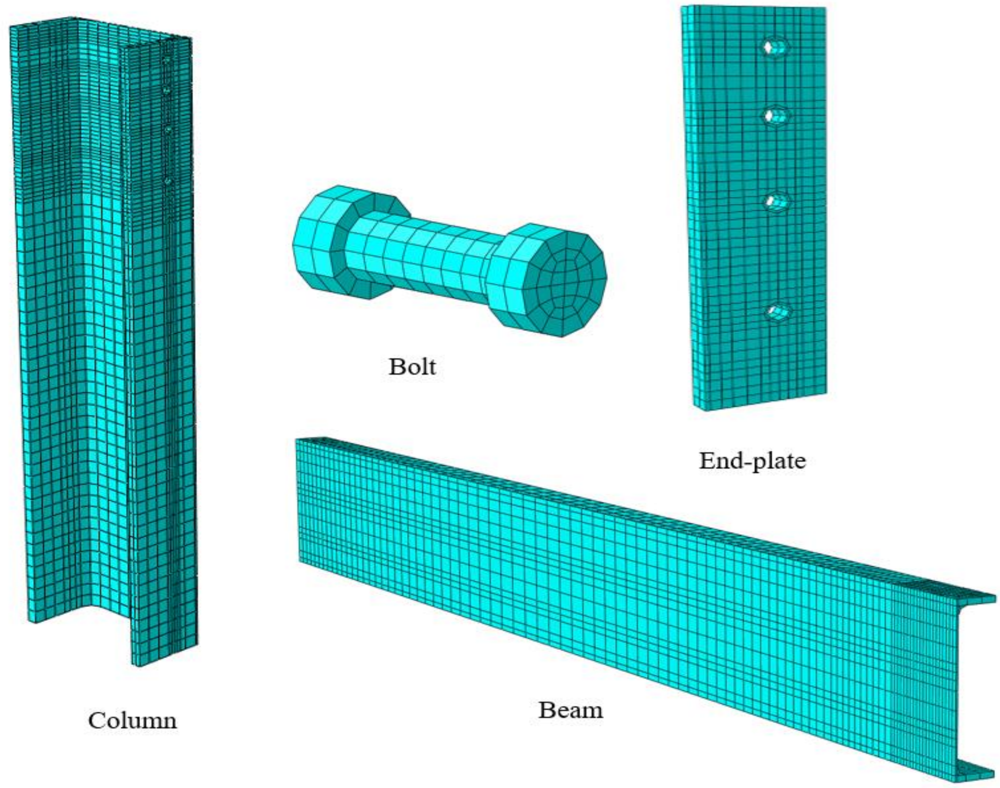


Figure 3:6 FE Mesh of the model

Eight-node brick elements with reduced integration (C3D8R) were used for all parts in the model. Reduced integration involves the elements having just a single integration point located at the element's centroid. The mesh was carefully symmetrically defined in critical zones, such as the regions around the bolt holes where likely failures would initiate. Besides, for the contact between contact elements, the meshing was fine enough for each element's node of the master surface to face a corresponding node of the slave surface elements.

CHAPTER 4 RESULTS AND RECOMMENDATIONS

4.1 Introduction

This chapter aims to validate the numerical model developed in Chapter 3, the predicted FE results are compared against the results obtained from the components method. The moment-rotation curve proposed by EC 03 for joints classification (see Figure 4:1) is used to validate the proposed FE model.

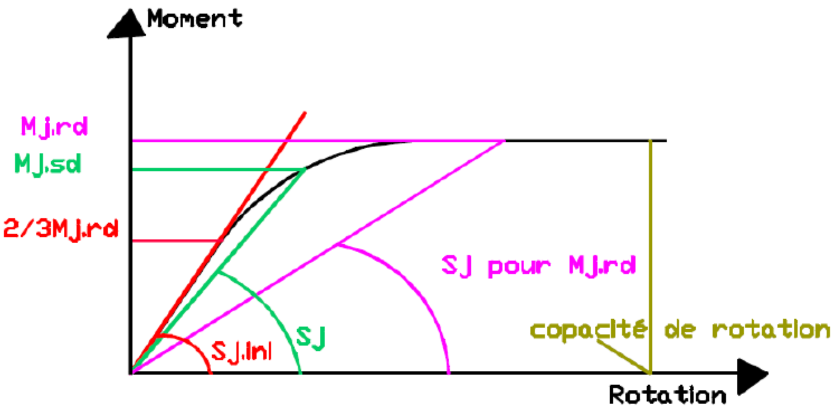


Figure II-6: Courbe M- non- linéaire.

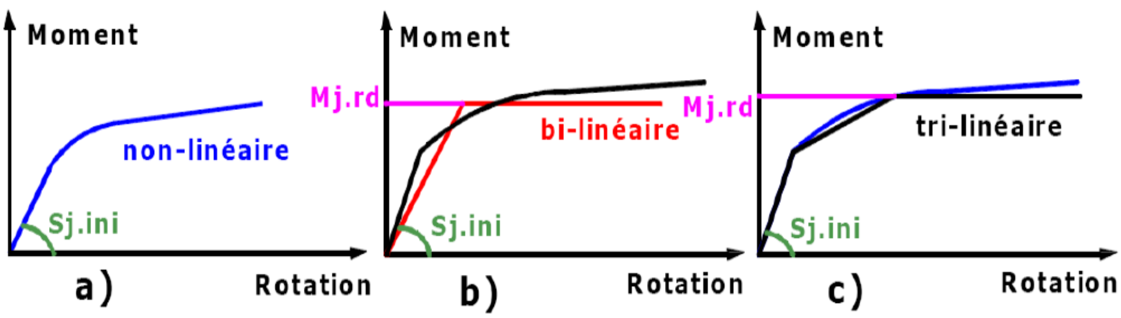


Figure 4:1 Idealizations of moment-rotation curves

4.2 Application of The Component Method

4.2.1 preliminary calculations

- Column:

$$M_{c.Rd(class1)} = \frac{W_{pl.y} f_{yb}}{y_{m0}} = \frac{2149 \times 235 \times 10^3}{1} = 505015000 N.mm = 505.015 KN.m \quad (4:1)$$

$$d_{wc} = h_c - 2t_{fc} - 2r_c = 320 - (2 \times 21) - (2 \times 27) = 224mm \quad (4:2)$$

$$m_{fc} = \frac{w - t_{wc}}{2} - 0.8r_c = \frac{120 - 12}{2} - 0.8 \times 27 = 32.4mm \quad (4:3)$$

$$e_{fc} = \frac{b_c - w}{2} = \frac{300 - 120}{2} = 90mm \quad (4:4)$$

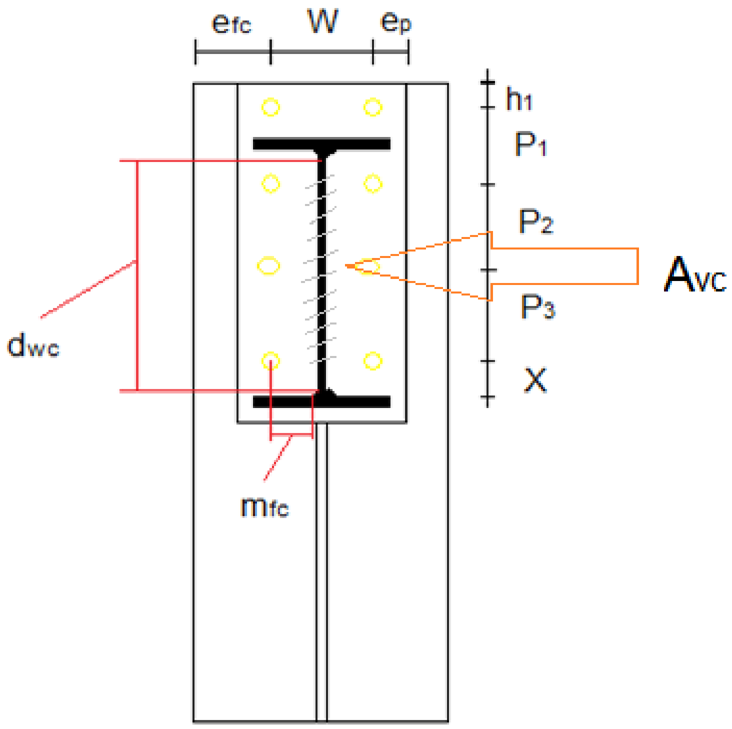


Figure 4:2 Geometrical properties for the column preliminary calculation

- Beam:

$$M_{b.pl.Rd(class1)} = \frac{W_{pl.y} f_{yb}}{y_{m0}} = \frac{804.10^3 \times 235}{1} = 188940000 N.mm = 188.94 KN.m \quad (4:5)$$

- End-plate

$$m_{p1} = \frac{w - t_{wb}}{2} - 0.8\sqrt{2}\alpha_w = \frac{120-8}{2} - 0.8\sqrt{2} \times 8 = 96.949mm \quad (4:6)$$

$$m_{p2} = (h_1 + P_1) - 69 - t_{fb} - 0.8\sqrt{2} \times 8 = 19.949mm \quad (4:7)$$

$$e_p = \frac{b_p - w}{2} = \frac{200-120}{2} = 40mm \quad (4:8)$$

$$\lambda_1 = \frac{m_{p1}}{m_{p1} + e_p} = \frac{96.949}{96.949+40} = 0.7079 \quad (4:9)$$

$$\lambda_2 = \frac{m_{p2}}{m_{p1} + e_p} = \frac{19.949}{96.949+40} = 0.1457 \quad (4:10)$$

$\alpha = 4.5$ → (see Appendix D)

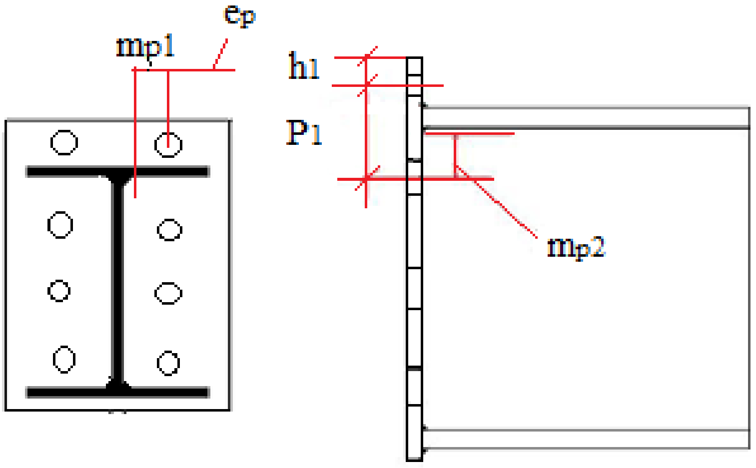


Figure 4:3 Geometrical properties for the end-plate preliminary calculation

- **Bolt**

$$F_{t.Rd} = \frac{0.9A_s f_{ub}}{y_{m2}} = \frac{0.9 \times 157 \times 800}{1.25} = 90432N \quad (4:11)$$

$$d_w = 26.75 \rightarrow e_w = \frac{d_w}{4} = \frac{26.75}{4} = 6.6875mm \quad (4:12)$$

$$L_b = t_{fc} + t_p + 0.5(h_h + h_n) = 21 + 20 + 0.5(10 + 13) = 57.5mm \quad (4:13)$$

4.3 Calculation the Resistance *F* and Stiffness *k* of Components

4.3.1 Component N°01: Column web in shear

$$F_{Rd,1} = \frac{0.9A_{vc} f_{y,wc}}{\sqrt{3} y_{m0}} = \frac{0.9 \times 4916 \times 235}{\sqrt{3} \times 1} = 600290.705N \quad (4:14)$$

$$Z_{eq} = h_b - \frac{t_{fb}}{2} - p_1 = 330 - \frac{12}{2} - 70 = 254mm \quad (4:15)$$

- Stiffness influence coefficient

β=1 → (see Appendix E)

$$k_{1(1)} = \frac{0.38 \times A_{VC}}{\beta z_{eq}} = \frac{0.38 \times 4916}{1 \times 254} = 7.355mm \quad (4:16)$$

→ $F_{Rd.1(1)} = 600kN; K_{1;(1)} = 7.355mm$

4.3.2 Component N°02: Column web in compression

$$b_{eff,c,wc} = t_{fb} + 2\sqrt{2a_f} + 5(t_{fc} + r_c) + s_p = 12 + 2\sqrt{2} \times 8 + 5 \times (21 + 27) + 20 = 294.63mm \quad (4:17)$$

$$\rightarrow \lambda = 1$$

$$w_1 = \frac{1}{\sqrt{1 + 1.3(b_{eff} \frac{t_{wc}}{A_{vc}})^2}} = \frac{1}{\sqrt{1 + 1.3(294.63 \frac{12}{4916})^2}} = 0.773 \quad (4:18)$$

The Factor ρ which takes into account the buckling of the column web in compression is calculated as follows:

$$if: \lambda_p \le 0.72 \rightarrow \rho = 1$$
$$if: \lambda_p \ge 0.72 \rightarrow \rho = \frac{\lambda_p - 0.2}{\lambda_p^{\ 2}} \qquad (4:19)$$

$$\lambda_p = 0.932\sqrt{\frac{b_{eff,c} d_{wc} f_{y,wc}}{Et^2_{wc}}} = 0.932\sqrt{\frac{294.63 \times 224 \times 235}{210000 \times 12^2}} = 0.697 \le 0.72 \qquad (4:20)$$

$$\Rightarrow so: \rho = 1$$

$$F_{Rd,2,(1)} = \frac{w_c b_{eff,c} t_{wc} f_{y,wc} \rho}{\gamma_{m0}} = \frac{0.773 \times 294.63 \times 12 \times 235 \times 1}{1} = 642252.152N \qquad (4:21)$$

Stiffness influence coefficient

$$k_{2(1)} = \frac{0.7 b_{eff,c} \times t_{wC}}{d_{wc}} = \frac{0.7 \times 294.63 \times 12}{224} = 11.05mm \qquad (4:22)$$

→ $F_{Rd.2(1)} = 642kN; K_{2;(1)} = 11.05mm$

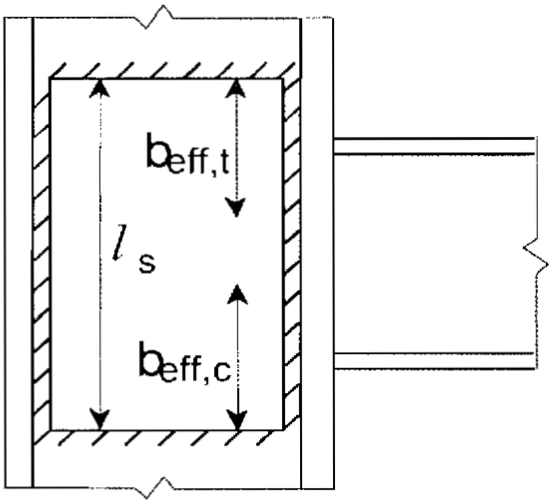


Figure 4:4 Effective width of the T-stub flange in compression and tension

4.3.3 Component N°03: Column web in tension

$$m_{wc} = \frac{w - t_{wc}}{2} - 0.8 r_c = \frac{120 - 12}{2} - 0.8 \times 27 = 32.4 mm \quad (4:23)$$

$$e_{wc} = \min\left(\frac{b_c - w}{2}; \frac{b_p - w}{2}\right) = \min\left(\frac{300 - 120}{2}; \frac{200 - 120}{2}\right) = 40 mm \quad (4:24)$$

$$L_{eff} = \min(4m + 1.25e; 2\pi m) = \min(4 \times 32.4 + 1.25 \times 40; 2 \times 3.14 \times 32.4)$$
$$= (179.6; 203.472) = 179.6 mm \quad (4:25)$$

$$\rightarrow \beta = 1 .... so$$

$$w_t = \frac{1}{\sqrt{1 + 1.3\left(b_{eff} \times \frac{t_{wc}}{A_{vc}}\right)^2}} = \frac{1}{\sqrt{1 + 1.3\left(294.63 \times \frac{12}{4916}\right)^2}} = 0.773 \quad (4:26)$$

$$F_{Rd,3.(1)} = \frac{w_t b_{eff.c} t_{wc} f_{y.wc}}{\gamma_{m0}} = \frac{0.773 \times 294.63 \times 12 \times 235}{1} = 642252.15 N \quad (4:27)$$

- Stiffness influence coefficient

$$K_{3.(1)} = \frac{0.7 b_{eff} t_{wc}}{d_{wc}} = \frac{0.7 \times 294.63 \times 12}{224} = 11.049 mm \quad (4:28)$$

$$\rightarrow \boxed{F_{Rd.3.(1)} = 642 kN; K_{3.(1)} = 11.049 mm}$$

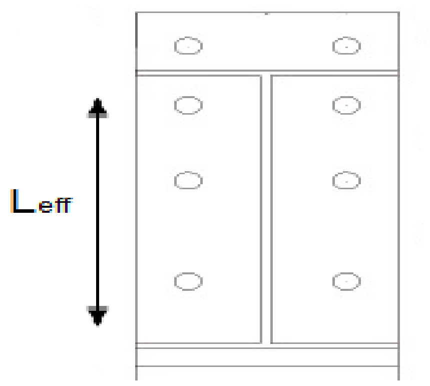


Figure 4:5 Effective length of the column web

4.3.4 Component N°04: column flange in bending

$$L_{eff.t.fc} = \min(4m + 1.25e; 2\pi m) = \min(4\times 32.4 + 1.25\times 40; 2\times 3.14\times 32.4) = 179.61mm \quad (4:29)$$

$$n = \min(e_{fc;} 1.25m_{fc}; e_p) = \min(90; 40.5; 40) = 40mm \Rightarrow K_{fc} = 1 \quad (4:30)$$

$$M_{pl.Rd} = \frac{0.25b_{eff.t.wc}t_{fc}^2 f_y}{y_{m0}} = \frac{0.25\times 294.63\times 21^2\times 235}{1} = 7633495.013N.m \quad (4:31)$$

$$F_{fc,Rd1} = \frac{4M_{pl,Rd}}{m} = \frac{4\times 7633495.013}{32.4} = 942406.79N \quad (4:32)$$

$$F_{fc,Rd2} = \frac{2M_{pl,Rd} + n\sum B_{t,Rd}}{m+n} = \frac{2\times 7633495.013 + 2\times 90432\times 40}{32.4+40} = 310794.89N \quad (4:33)$$

$$F_{Rd,4(1)} = \min(F_{fc,Rd1}; F_{fc,Rd2}) = 310794.89N \quad (4:34)$$

- Stiffness influence coefficient

$$K_{4.(1)} = \frac{0.9l_{eff.t.fc}t_{fc}^3}{m^3_{fc}} = \frac{0.9\times 179.61\times 21^3}{32.4^3} = 44.0145mm \quad (4:35)$$

→ $F_{Rd.4.(1)} = 311kN; K_4 = 44.0145mm$

4.3.5 Component N°05: End-plate in bending

$$l_{eff} = min(\alpha m; 2\pi m) = min(4.5\times 96.949; 2\times 3.14\times 96.949)$$
$$= \min(436.27; 608.84) = 436.27mm \quad (4:36)$$

$$M_{pl,Rd} = \frac{0.25l_{eff,wc}t_p^2 f_y}{y_{m0}} = \frac{0.25\times 436.27\times 20^2\times 235}{1} = 10252.345N \quad (4:37)$$

$$F_{eq,Rd.1} = \frac{4M_{pl,Rd}}{m_{p1}} = \frac{4\times 10252.345}{96.949} = 422.999N \quad (4:38)$$

$$F_{eq,Rd.2} = \frac{2M_{pl,Rd} + n\sum F_{t,Rd}}{m_p + n_p} = \frac{2\times 10252.345 + 40\times 4\times 90432}{96.949+40} = 105803.07N \quad (4:39)$$

$$F_{Rd,5,(1)} = \min(F_{eq,Rd.1}, F_{eq,Rd.2}) = 105.803N \qquad (4:40)$$

- Stiffness influence coefficients

$$K_{5.(1)} = \frac{0.9 l_{eff} t_p{}^3}{m_{p1}{}^3} = \frac{0.9 \times 436.27 \times 20^3}{96.949^3} = 3.447mm \qquad (4:41)$$

→ $F_{Rd,5(1)} = 10kN; K_{5,(1)} = 3.447mm$

4.3.6 Component N°06: Beam web in tension

$$b_{eff,tw,b} = l_{eff,p} = 436.27mm$$

$$F_{Rd,6,(1)} = b_{eff,tw,b} t_{wb} \frac{f_{y,wb}}{y_{m0}} = 436.27 \times 20 \times \frac{235}{1} = 2050469N \qquad (4:42)$$

- Stiffness influence coefficient

$$K_{6,(1)} = \infty$$

→ $F_{Rd,6(1)} = 2050kN; K_{6,(1)} = \infty$

4.3.7 Component N°07: Beam web and flange in compression

$$F_{Rd.7(1)} = \frac{M_{b.Rd}}{h_b - t_{fb}} = \frac{505015000}{330 - 12} = 1588097.484N \qquad (4:43)$$

- Stiffness influence coefficient

$$K_{7.(1)} = \infty$$

→ $F_{Rd,7(1)} = 1588kN; K_{7,(1)} = \infty$

4.3.8 Component N°09: bolts in tension

$$F_{Rd,9,(1)} = 2 \times B_{t,Rd} = 2 \times 90432 = 180864N \qquad (4:44)$$

- Stiffness influence coefficients

$$F_{c,Rd(1)} = \min\left(F_{Rd,2(1)}; F_{Rd,7(1)}\right) = \min\left(642.252; 1588.097\right) = 642.252KN \qquad (4:45)$$

$$K_{9.(1)} = 1.6\frac{A_{s.b^3}}{L_b} = 1.6 \times \frac{157}{57.5} = 4.369mm \qquad (4:46)$$

→ $F_{Rd,9(1)} = 181kN; K_{9,(1)} = 4.369mm$

4.4 Assembly of Components

- Design compression resistance of the joint:

$$F_{c,Rd(1)} = \min\left(F_{Rd,2(1)}; F_{Rd,7(1)}\right) = \min\left(642.252; 1588.097\right) = 642.252KN \qquad (4:47)$$

- Stiffness coefficient for the tension zone:

$$K_{t,(1)} = \frac{1}{\frac{1}{K_3} + \frac{1}{K_4} + \frac{1}{K_5} + \frac{1}{K_8} + \frac{1}{K_9}} = \frac{1}{\frac{1}{11.049} + \frac{1}{44.014} + \frac{1}{3.447} + \frac{1}{\infty} + \frac{1}{4.369}} \qquad (4:48)$$
$$= \frac{1}{0.091 + 0.023 + 0.29 + 0 + 0.229} \Rightarrow K_{t,(1)} = 1.5798mm$$

- Design tension resistance of the joint:

$$F_{t,Rd,(1)} = min\left(F_{Rd,3(1)}; F_{R4,(1)}; F_{Rd5,(1)}; F_{Rd6,(1)}; F_{Rd9,(1)}\right) \qquad (4:49)$$
$$= \min\left(642.252; 310.79; 180.864; 2050.47; 105.803\right)$$

$$\Rightarrow F_{t,Rd,(1)} = 106kN$$

- Design resistance of the joint:

$$F_{Rd,(1)} = \min\left(F_{t,Rd,(1)}; F_{c,Rd,(1)}; F_{Rd,1(1)}\right) = \min\left(105.803; 642.252; 600.290\right) \qquad (4:50)$$

$$\Rightarrow F_{Rd,(1)} = 106kN$$

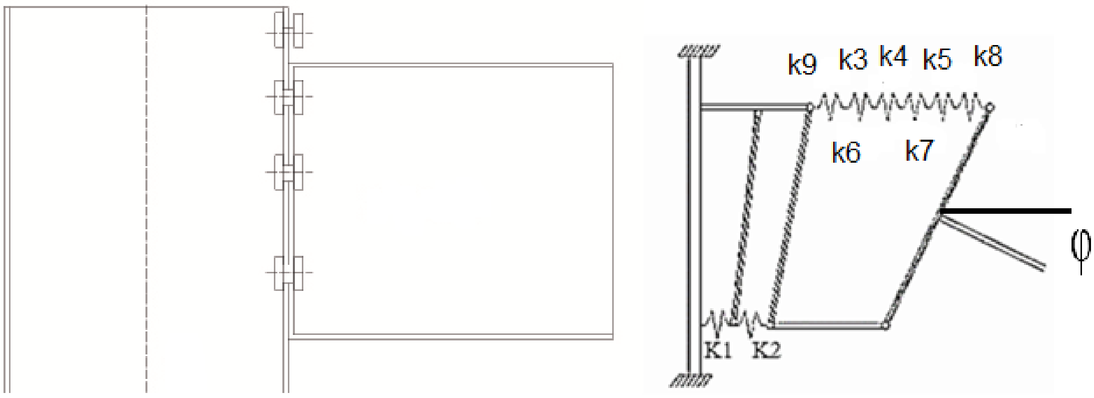


Figure 4:6 Mechanical model with springs of the different components

- Design moment resistance of the joint:

$$M_{j,Rd} = F_{Rd,(1)} \times z = 105.803 \times 254 \Rightarrow M_{j,Rd} = 27kN \qquad (4:51)$$

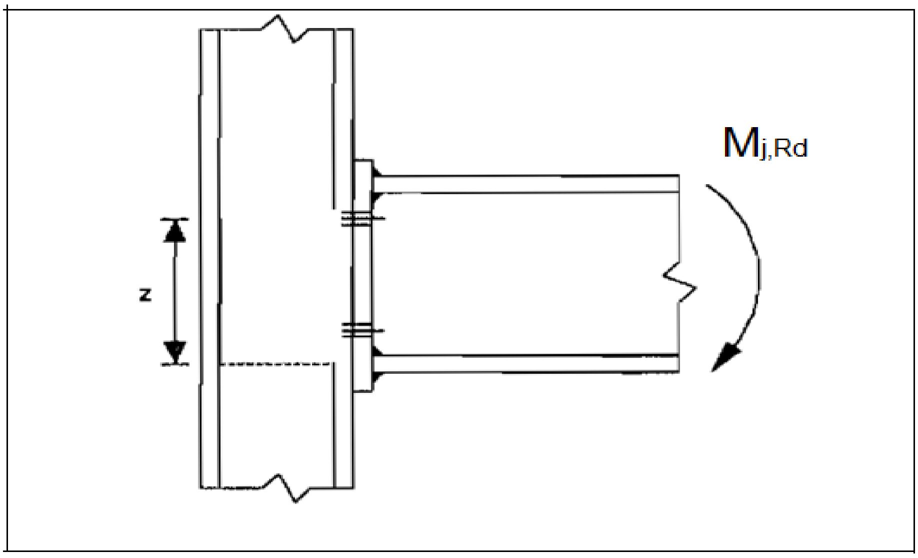


Figure 4:7 lever arm *z* and force distributions for deriving the design moment resistance $M_{j,Rd}$

- Design elastic moment of the joint:

$$M_{e,Rd} = \frac{2}{3} M_{j,Rd} = \frac{2}{3} \times 27 \Rightarrow M_{e,Rd} = 18kN \tag{4:52}$$

4.4.1.1 Initial rotational stiffness of the joint:

$$S_{j,ini} = \frac{E_\alpha Z_{eq}^{\ 2}}{\frac{1}{K_1} + \frac{1}{K_2} + \frac{1}{K_7} + \frac{1}{K_t}} = \frac{210000 \times (254)^2}{\frac{1}{7.355} + \frac{1}{11.05} + \frac{1}{\infty} + \frac{1}{1.5798}} = 15763.966kN.m \tag{4:53}$$

4.4.1.2 Rotational stiffness of the joint:

$S_j = \frac{S_{j,ini}}{\eta}$, from Appendix G → η = 2

$$S_j = \frac{S_{j,ini}}{2} = \frac{15763.966}{2} = 7882kN.m \tag{4:54}$$

4.5 **Classification of the Joint:**

4.5.1 **Classification by stiffness:**

A beam-to-column connection can be classified as rigid, pinned or semi-rigid depending on its stiffness by determining its initial rotational stiffness $S_{j,ini}$ and comparing it with the classification limits.

The frame is considered unbraced:

$$0.5 \times \frac{EI_b}{L_b} = \frac{0.5 \times 2.1 \times 11766.9}{3} = 4118.415KN.m \tag{4:55}$$

$$25 \times \frac{EI_b}{L_b} = \frac{25 \times 2.1 \times 11766.9}{3} = 205920.75KN.m \tag{4:56}$$

$$\Rightarrow 0.5\ EI/L\ <\ Sj,ini\ <\ 25\ EI/L$$

$$\rightarrow 4118\ <\ 15764 <\ 205921$$

Depending on the configuration of the structure (length of the beam, unbraced mode), the joint is classified as semi-rigid

4.5.2 Classification by strength:

The resistance calculated for the weakest of the assembled elements is $F_{4,\,Rd}$ (column flange).

$$M_{c.Rd} = \frac{W_{pl.y}\, f_{yb}}{y_{m0}} = \frac{2149 \times 235 \times 10^{3}}{1} = 505015000 N.mm = 505.015 kN.m \quad (4:57)$$

→ $M_{j,Rd} = 27 kN.m$

According to the classification limits given in section4.3.2

$$0.25 M_{c.pl,Rd} = 126.25 kN.m$$
$$\Rightarrow So : M_{j,Rd} \leq 0.25\, M_{full-strength}$$

⇨ The joint is classified **as articulated**

4.6 Validation of the FE Model

The proposed FE model's validity is examined by comparing the co-obtained numerical results with those determined analytically by the component method of the EC03 [5]. In this work an end-plate steel joint was designed under a static load to predict the structural behaviour (Mj, Rd, φj and Sj,ini). The details of the investigated connection are shown in Figure 3:2 and Table 3:2. The numerical and analytical results are compared in terms of design moment resistance Mj,Rd and maximum rotation of the joint φj. The rotation was calculated using the approximation (4:58).

$$\varphi \approx \tan\varphi = \frac{(u_1 - u_2)}{h_b} \quad (4:58)$$

where:

φ is the rotation;

u_1 is the displacement of the beam flange in tension (upper flange);

u_2 is the displacement of the beam flange in compression (lower flange);

h_b is the beam height.

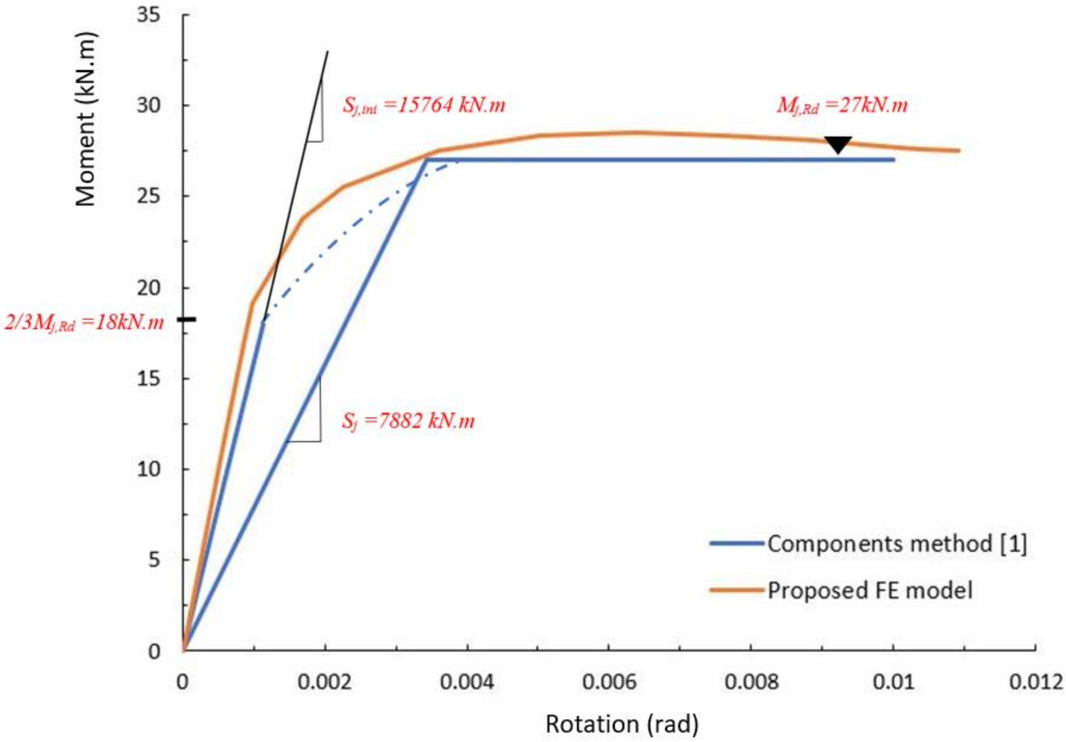


Figure 4:8 Comparison between the predicted moment-rotation curve against the idealization curve of the component method [1].

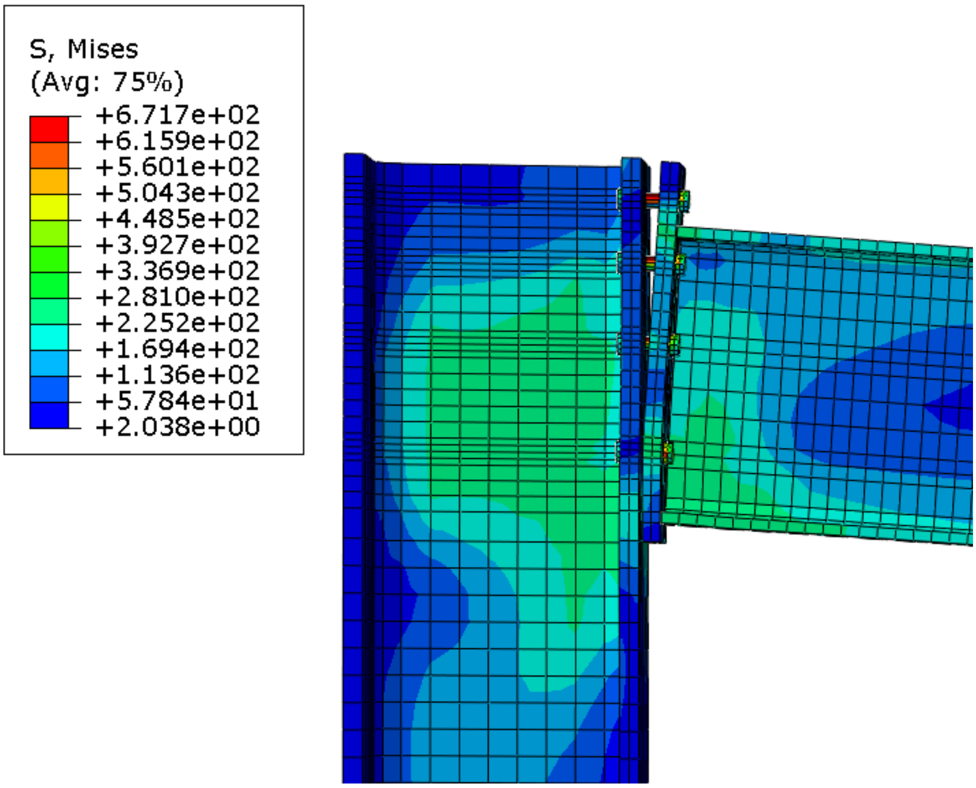


Figure 4:9 Deformed model and Von-Mises stress distribution

Table 4:1 Comparison of results for both approaches (FEM and EC03)

	$M_{j,Rd}$ (kN.m)	ϕ_{jini} (rad)	S_j (kN.m)	$S_{j,ini}$ (kN.m)	$M_{E,Rd}$ (kN.m)
FE method	28.5	0.000983	7638	16515	19
Components method [1]	27	0.00171	7882	15764	18

Comparing the moment versus rotation curves as reported in Figure 4:8 demonstrates that finite element results are highly correlated with analytical results of the component method [1]. Finally, the FE model developed herein is capable of accurately predicting the behavior of beam-to-column steel joints. In particular, as reflected in Table 4:1.Figure 4:9 shows an image of the finite element model with its deformed geometry.

4.7 **Stress Concentration**

Stress concentrations were observed from the stress contour plot obtained from the FE model that a high-stress concentration was developed in the junction between the end-plate and the bottom beam flange. This phenomenon is a common occurrence in FEA; it is a place of the mesh where the stress raises above the applied nominal stress, and the stress will converge towards a finite value, given that the mesh is sufficiently refined (see Figure 4:10).

A refined FE mesh was adopted for the end-plate and the bottom beam flange junction region to reduce stress concentration.

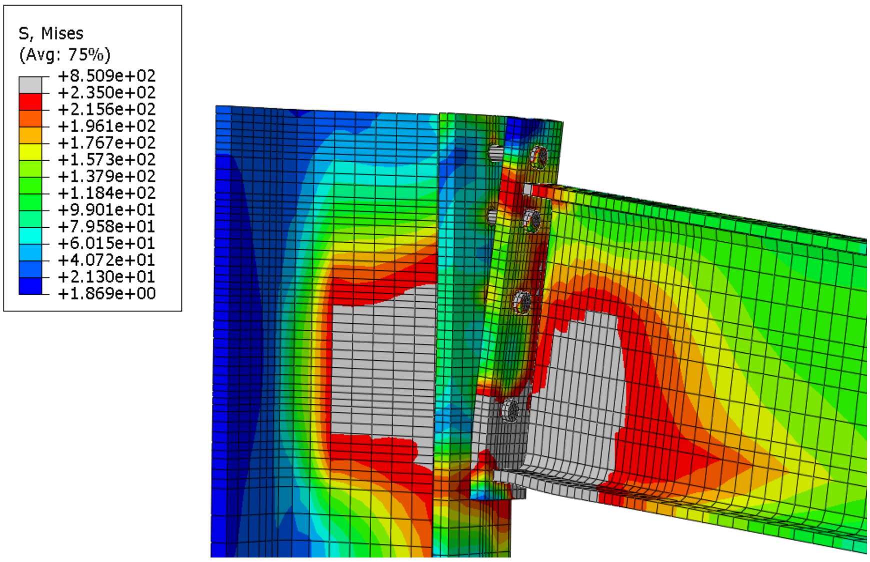


Figure 4:10 Stress concentration in the junction between end plate and bottom beam flange.

4.8 Conclusion

The thesis's main purpose is to study the behaviour of metallic structures with semi-rigid, bolted assemblies and use angles as a configuration., which is done under the effect of a response spectrum and different stiffness models to determine the main characteristics of these assemblies ($M_{j,Rd}$, φ_j and $S_{j,ini}$).

The purpose of this is to highlight the various parameters influencing the overall behaviour of the structures. For this, two approaches were used. The first was devoted to theoretical research. Importance was given to several prediction equations that model the moment-rotation behaviour of semi-rigid assemblies. The research was based on assembly models that adequately represent the actual behaviour of assemblies. These models were based on parameters that can be determined easily from the assembly configuration and their detail.

Current code requirements and recommendations focus on the study of joints as the primary sources of inelastic behaviour of the structure. However, very few works provide a model that describes the real behaviour of the assemblies, except the results of the experimental tests, which remain expensive.

Research that deals with assemblies with the FEM is generally limited to simple finite elements (beam element, for example) and assemblies where the behaviour is relatively well understood, and the simplicity of the modelling is evident.

CHAPTER 5 CONCLUSIONS

5.1 SUMMARY

The thesis's work involved developing a FE model of a semi-rigid end-plate connection and ensure its accuracy to predict the structural behaviour of this type of connections such as, the design moment resistance $M_{j,Rd}$, the initial rotational stiffness $S_{j,ini}$ and the rotation of the joint φ.

It is known that numerical simulations are very important to understand the actual behaviour of engineering systems better. The ultimate advantages of FE analyses are their cost-effective nature and the possibility of investigating difficult (if not impossible) variables to be measured and/or monitored during tests.

To obtain a reliable model that can be effectively utilized in engineering systems, the engineers must be familiarised with the techniques involved in the numerical process and the range of tools that the commercial codes offer. In addition, a critical analysis of the results is always advised, as it will increase the model's reliability.

An end-plate steel connection design and its numerical simulation have been presented. Finite element analysis was used in the investigation. The FE model incorporated the material properties of the different components of the composite beam and the composite joint. The component method was provided to evaluate the structural behaviour of the end-plate steel joint; a finite element model is developed using ABAQUS software to determine the moment-rotation curve. The idealization curves (see Figure 4:1) of the Eurocode 03 [1] were used to validate the numerical results.

From this research, the following conclusions were drawn:

1) The finite element model presented in this thesis predicted the behaviour of the end-plate steel joint accurately. Based on the FE model, it was found that the FE predict well the initial joint stiffness ($S_{j,ini}$), the rotation of the joint (φ) and the design moment resistance ($M_{j,Rd}$).
2) The FE model has almost the same moment-rotation curve comparing the one obtained from the component method.

5.2 Final Considerations

Further studies are required to explore the applicability of the proposed FE model to other beam-to-column configurations and better understand the behaviour of steel joints, which constitutes a perspective to the presented work.

As a perspective, we propose to continue our research on this issue by exploring the following points:

- Conduct experimental studies to examine the applicability of the proposed model with other configurations of steel joints and different types of loading (. i.e. cycling load, seismic load, etc.).
- Perform a FE analysis based on the proposed numerical model to investigate the most important parameters affecting the structural behaviour of a steel beam-to-column joint. The applicability of the proposed FE model on such connections will be verified through comparative studies.

Finally, from a personal point of view, this Master thesis provided a chance for us to use ABAQUS software and to improve our knowledge in the field of steel structures.

REFERENCES

[1] E. CEN, 1-8 Eurocode 3: Design of Steel Structures–Part 1–8: Design of Joints, Brussels: European Committee for Standardization, 2005.

[2] A. I. S¸TEFANCU, I. OLTEANU and S. C. MELENCIUC, "SEISMIC DESIGN OF STEEL CONNECTIONS," *Bulletin of the Polytechnic, Institute of Jassy,* January 2009.

[3] E. R. Romero, Master of science thesis : Finite element simulation of a bolted steel joint in fire using abaqus program, Finland: Tampere University of Technology , 2010.

[4] ABAQUS, Theory and User's Manuals, Dassault Systèmes, S. I. M. U. L. I. A. (2014), 2014.

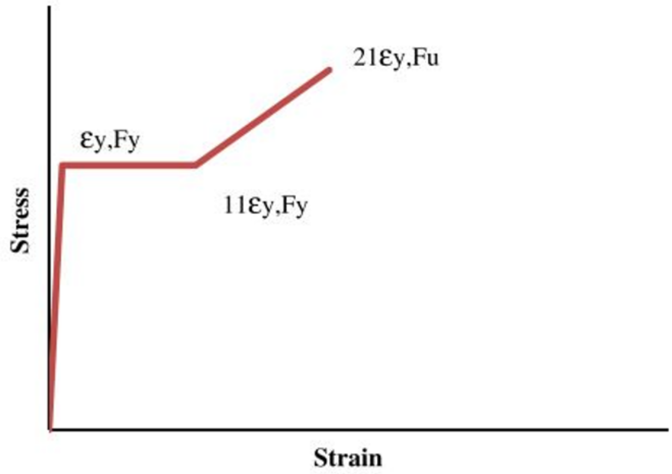
[5] A. Bahaz, Caractérisation des Assemblages Semi-rigides Poteau – Poutre par la Méthode des Composantes, Laghouat: University of Laghouat, 2011.

[6] E. C. f. Standardization, Eurocode 3: Design of steel structures—Part 1.1: General rules and rules for buildings, 2005.

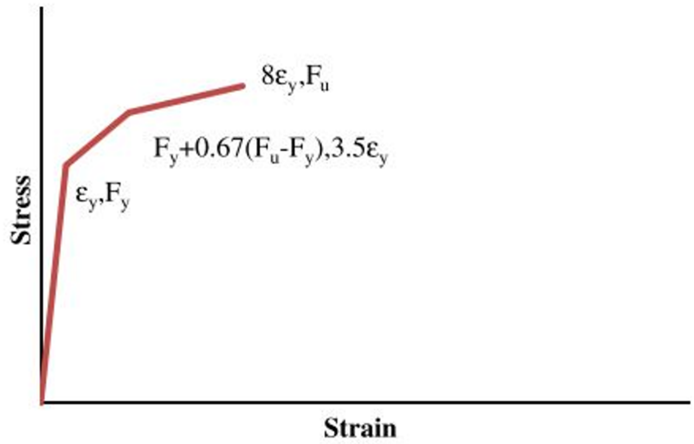
[7] H. B. I. W. D. J. B. &. P. R. J. Yu, "Numerical simulation of bolted steel connections in fire using explicit dynamic analysis," *Journal of Constructional Steel Research,* pp. 64(5), 515-525, 2008.

[8] M. B. I. W. D. J. B. &. P. R. J. Sarraj, "Finite element modelling of steel fin plate connections in fire," *Fire Safety Journal,* pp. 42(6-7), 408-415, 2007.

APPENDIX



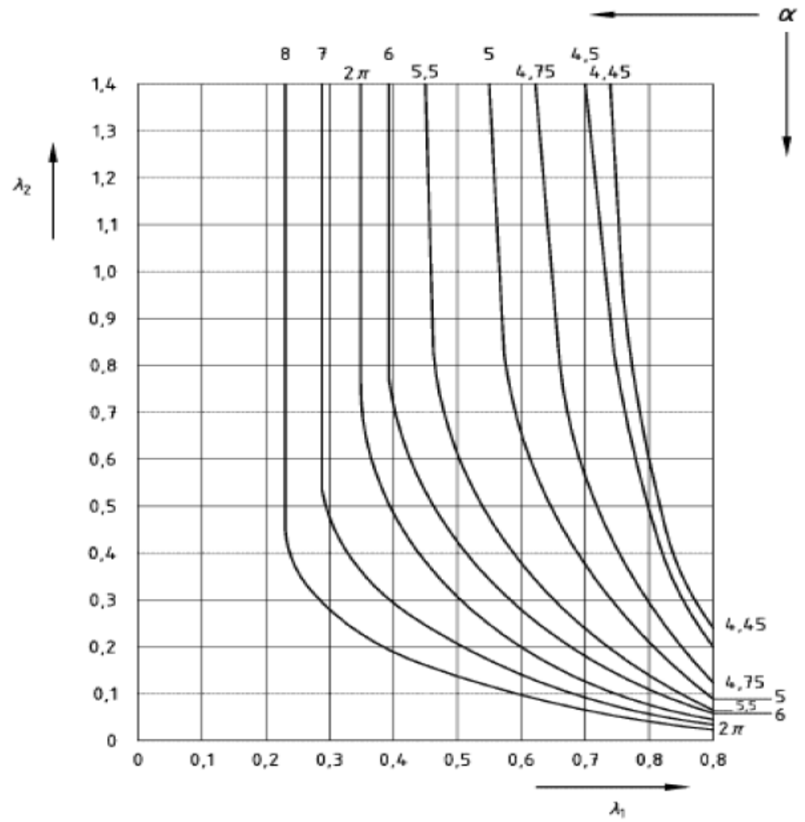
Appendix A Stress-strain curve for beam, column and end-plate material.



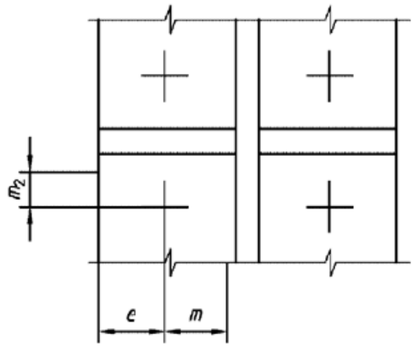
Appendix B Stress-strain curve for bolts material.

E	210000 MPa
Poisson's Ratio	0.3
Plastic behaviour of the structural steel (Column, beam and end-plate)	
235.26	0
237.89	0.01112
445.22	0.02211
Plastic behaviour of the bolts	
662.07	0
782.41	0.0078
850.87	0.02169

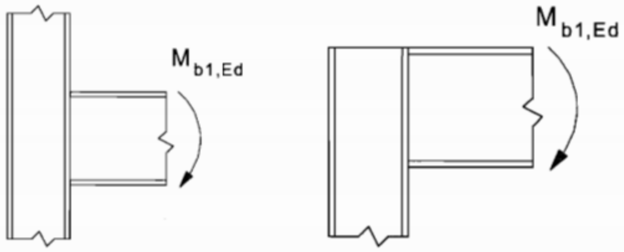
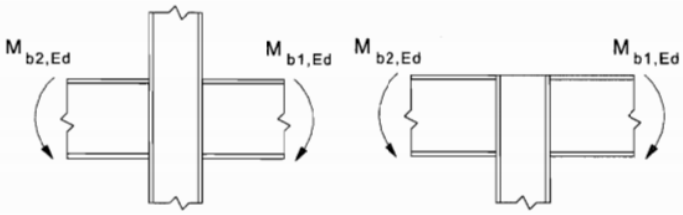
Appendix C Structural steel material properties



$$\lambda_2 = \frac{m_{p2}}{m_{p1} + e} \; ; \; \lambda_1 = \frac{m_{p1}}{m_{p1} + e}$$



Appendix D Values of α for stiffened column flange and end-plates

Type of joint configuration	Action	Value of β
	$M_{b1,Ed}$	$\beta \approx 1$
	$M_{b1,Ed} = M_{b2,Ed}$	$\beta = 0$ *)
	$M_{b1,Ed} / M_{b2,Ed} > 0$	$\beta \approx 1$
	$M_{b1,Ed} / M_{b2,Ed} < 0$	$\beta \approx 2$
	$M_{b1,Ed} + M_{b2,Ed} = 0$	$\beta \approx 2$
*) In this case the value of β is the exact value rather than an approximation.		

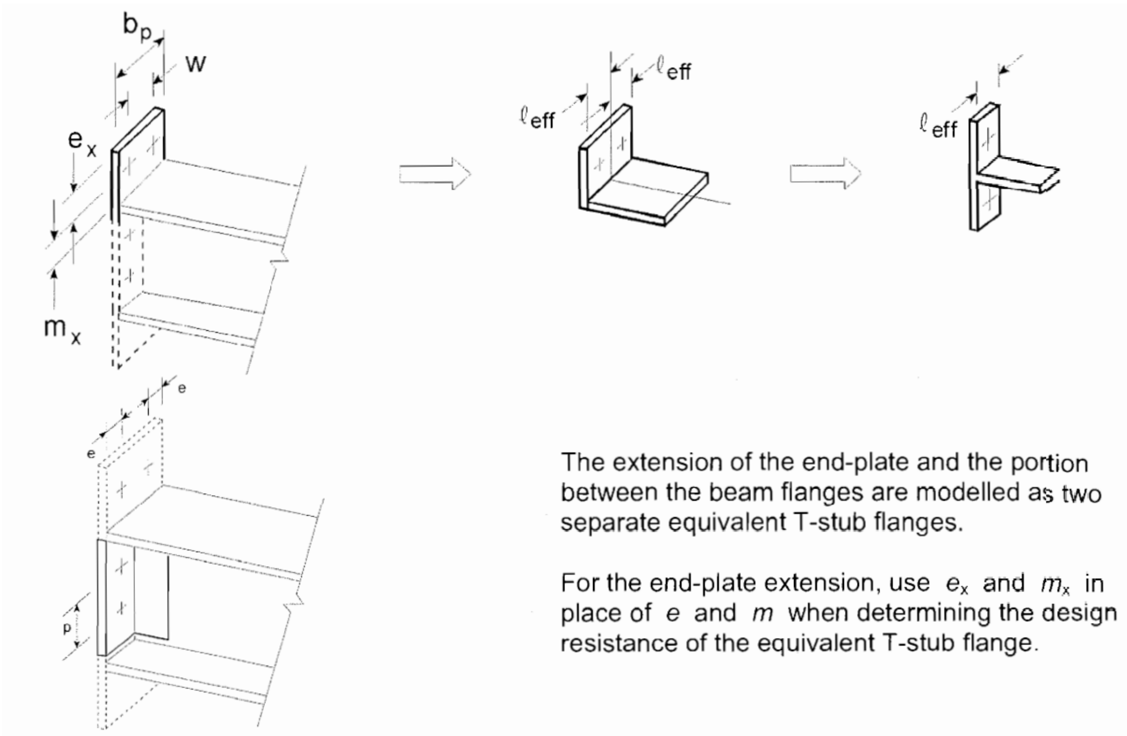
Appendix E Approximate values for the transformation parameter **β**

Transformation parameter β	Reduction factor ω
$0 \leq \beta \leq 0{,}5$	$\omega = 1$
$0{,}5 < \beta < 1$	$\omega = \omega_1 + 2(1-\beta)(1-\omega_1)$
$\beta = 1$	$\omega = \omega_1$
$1 < \beta < 2$	$\omega = \omega_1 + (\beta - 1)(\omega_2 - \omega_1)$
$\beta = 2$	$\omega = \omega_2$
$\omega_1 = \dfrac{1}{\sqrt{1 + 1{,}3(b_{eff,c,wc}\, t_{wc} / A_{vc})^2}}$	$\omega_2 = \dfrac{1}{\sqrt{1 + 5{,}2(b_{eff,c,wc}\, t_{wc} / A_{vc})^2}}$

Appendix F Reduction factor **ω** for shear interaction

Type of connection	Beam-to-column joints	Other types of joints (beam-to-beam joints, beam splices, column base joints)
Welded	2	3
Bolted end-plates	2	3
Bolted flange cleats	2	3,5
Base plates	-	3

Appendix G Stiffness modification coefficient



Appendix H Modelling an extended end-plate as separate T -stubs

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	πm + 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 $\ell_{eff,1} \leq \ell_{eff,cp}$		$\sum\ell_{eff,1} = \sum\ell_{eff,nc}$ but $\sum\ell_{eff,1} \leq \sum\ell_{eff,cp}$	
Mode 2:	$\ell_{eff,2} = \ell_{eff,nc}$		$\sum\ell_{eff,2} = \sum\ell_{eff,nc}$	

Appendix I Effective lengths for an end-plate