Volume 5, Number 4 (August, 2012) p. 468-499 • ISSN 1983-4195

IBRACON

REVISTA IBRACON DE ESTRUTURAS E MATERIAIS
IBRACON STRUCTURES AND MATERIALS JOURNAL

Parametric study on the behaviour of bolted composite connections

Estudo paramétrico do comportamento de ligações mistas parafusadas

M. N. KATAOKA [a]
kataoka@sc.usp.br

A. L. H. C. EL DEBS [b]
analucia@sc.usp.br

Abstract

The studied connections are composed of concrete filled steel tubes (CFT) connected to composite beams by passing through bolts, endplates and steel deck, which also contributes to support the applied loads. The parametric analysis presented in this work is based on numerical simulations performed with software TNO Diana, using experimental results to calibrate the reference numerical model. The influence of three main parameters, being them the bolts diameter, the slab height and the beams cross section, was evaluated. According to the obtained bending moment versus rotation curves, it was concluded that, among the three parameters analyzed, the most important one was the bolts diameter. About the beams cross section, inconclusive results were achieved, probably due to the incompatibility between the 16 mm bolts and the robust beam cross sections considered in the parametric analysis.

Keywords: composite connection, numerical simulation, mixed column (CFT).

Resumo

A ligação em estudo é constituída por pilar misto preenchido com concreto ligado às vigas mistas por meio de parafusos passantes, chapa de topo e laje com forma de aço incorporada, que também contribui no combate aos esforços aplicados. O estudo paramétrico do presente trabalho é baseado em simulações numéricas realizadas no programa TNO Diana. O modelo numérico de ligação considerado como referência foi calibrado a partir de resultados experimentais e para o estudo do seu comportamento foram variados parâmetros importantes como o diâmetro dos parafusos, altura da laje e seção da viga. Como resultado das análises concluiu-se que dentre os três parâmetros analisados o que mais influenciou o comportamento da ligação, tendo como base a curva momento fletor versus rotação, foi o diâmetro dos parafusos, obtendo resultados inconclusivos no caso da seção da viga, pois seções mais robustas não foram compatíveis com os parafusos de 16 mm de diâmetro utilizados no modelo de ligação de referência.

Palavras-chave: ligação mista, simulação numérica, pilar misto.

[a] *Department of Structural Engineering, School of Engineering of São Carlos, University São Paulo, kataoka@sc.usp.br, Trabalhador Saocarlense Avenue, nº 400, CEP: 13566-580, São Carlos, SP, Brazil.*

[b] *Department of Structural Engineering, School of Engineering of São Carlos, University São Paulo, analucia@sc.usp.br, Trabalhador Saocarlense Avenue, nº 400, CEP: 13566-580, São Carlos, SP, Brazil.*

Received: 02 Dec 2011 • Accepted: 11 Abr 2012 • Available Online: 03 Aug 20122

1. Introduction

The technological development provided an advance in numerical analysis in order to describe the structures behaviour. One of the available tools is the finite element method (FEM) in association with failure criteria of considered materials that can be used to simulate the structural systems, especially composite structures. The program DIANA is a computational modeling tool that uses this method.

The numerical analysis has been increasingly used to study the behaviour of structures with some degree of nonlinearity, which was not possible before because of the lack of computational resources. Therefore, large structures that previously could only be studied through small-scale models, now can be analyzed with real dimensions, since the required resources nowadays are easily available.

In this context, the performance of numerical modeling as a complement to experimental analysis, or even to replace it when impracticable, has become increasingly common in academia, opinion shared by many authors [1, 2]. These researchers complement including the cost factor, because the substitution of physical models by numerical models exempts the final cost of the research.

During the review of the literature many studies about composite columns and numerical analysis were found. Most of the researches involving composite structures come from countries where the incidence with earthquakes is very common because of their good behaviour in such situation. An example is the research of [3], conducted in South Korea, which includes results of parametric analysis performed from tests and numerical simulations of connections involving composite columns filled with concrete and diaphragms.

Another study considered important is [4], which proposed a typology of bolted connection with circular cross section CFT column. In this paper it was presented an analytical study, using a refined three-dimensional model that provided an accurate understanding of the global behaviour of the connection, including the distribution of stresses on the contact surfaces. In Brazil there are not many researches and technical information about composite structures. This building system was introduced in this country around the 1950s, but its use did not grow because Brazil culture prefers reinforced concrete structures. Currently, studies of this building system are growing, such as in the Federal University of Minas Gerais, where studies about bolted connections between steel beam and composite column were found [5]. In the School of Engineering of Sao Carlos - USP a research group on composite structures has been developed, which can emphasize the research of [6], who conducted a theoretical and experimental study about filled columns submitted to axial compression and [7] wherein composite connections were analyzed.

The model of beam-column connection studied in this paper involves composite column filled with concrete which was bolted to the beams through endplates. The numerical analysis involves the variation of some structural parameters in the computer model. The computer model considered as reference for the parametric study was calibrated by experimental results. The structural parameters which influence was studied were bolts diameter, slab height and beam cross section. Each parameter was varied one at a time, being the other characteristics maintained equal to the reference model.

It was selected for this study a composite beam-column connection since the use of mixed or composite building systems expands the solutions in reinforced concrete and steel structures, producing architectural and economic benefits. Compared to the characteristics of reinforced concrete construction, the steel-concrete composite structure is competitive when it is used in structures with median and large spans. This composite building system is characterized by less time of execution and weight reduction, which provides economical foundations. The fire protection is another factor that influences the choice among reinforced concrete structures, steel structures and mixed structures, affecting significantly the final cost.

In prefabricated structures, the connections may not have the same stiffness of a monolithic structure; therefore the joints performance has a great importance. For a long time, structural analysis was performed considering the connections with rigid or pinned behaviour. With all researches performed and also with the practice on site, it was possible to demonstrate the inadequacy of classifying the connections only as rigid or pinned. The usual behaviour of connections is intermediate between these two idealized situations and to define this behaviour it must be use the term “semi-rigid connections.”

2. Design of the connection

In the physical model was used CFT column with square cross section and dimensions of 200x200 mm, the walls of the steel tube had a thickness of 8 mm and it was composed by two “U” profiles. The steel beams with “I” cross-section had 250 mm in height and 100 mm width, the flanges had 7.5 mm of thickness and the web 6.3 mm. All profiles were made with ASTM A-36 steel.

The connection was designed with eight bolts with 16 mm of diameter made with SAE 1020 steel and endplate with 22.2 mm of thickness made with ASTM A-36 steel. The column had 1950 mm in height and each beam had 1650 mm in length.

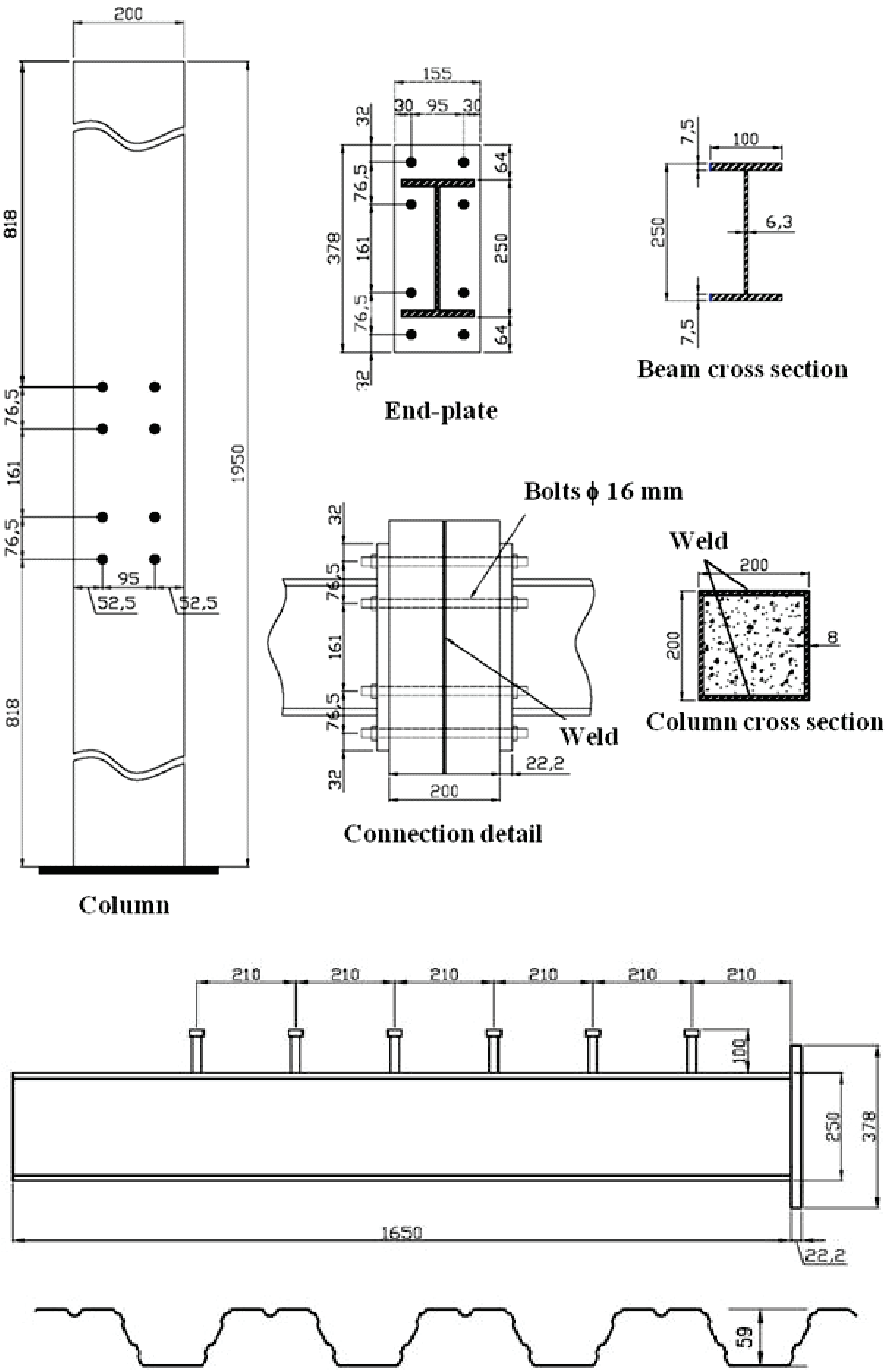
The shear connectors used to provide the joint action between the steel beam and the slab had a diameter of 19 mm, 100 mm of height and yielding stress of 415 MPa, according to the manufacturer›s information. In each beam were welded six shear connectors, the distance from each other was 210 mm (Figure 1).

The slab had 800 mm of width and 120 mm of height including 25 mm of concrete cover. The steel plates used in the slab had 0.80 mm of thickness and 59 mm of height. They were provided with 840 mm of width and 2500 mm of length and according to manufacturer’s specification, the steel plates weighing 9.14 kgf/m². The Figure 1 shows the complete design of the connection that was performed according to the specifications of ABNT NBR 8800:2008 [8] code.

The cruciform format was the final configuration of the connection model, simulating a central column connected to two cantilever beams. The dimensions of the set up model were 1950 mm in height and 3544.4 mm of length, as indicated in Figure 2.

As shown in Figure 3, in the slab reinforcement were used eight bars with 12.5 mm diameter in the longitudinal direction. In another direction were used twenty four bars with of 8.0 mm of diameter. The continuity bars were connected to the column with steel gloves in attempt to provide continuity to these bars and increase the connection stiffness (Figure 4). The steel area used in the longitudinal direction was 981.75 mm²

Figure 1 – Elements which are part of the connection (unit: mm)



3. Experimental program

The experimental program developed for this paper was performed at the Laboratory of Structures in Department of Structural Engineering in School of Engineering of Sao Carlos - USP. In this program the composite beam-column connection is subjected to cyclic loading with displacement reversals and the test was conducted in accordance with the representation in Figure 5.

The test results were used to calibrate the numerical model. Some important parameters were varied in the numerical model in order

Figure 2 – Dimensions of the tested connection

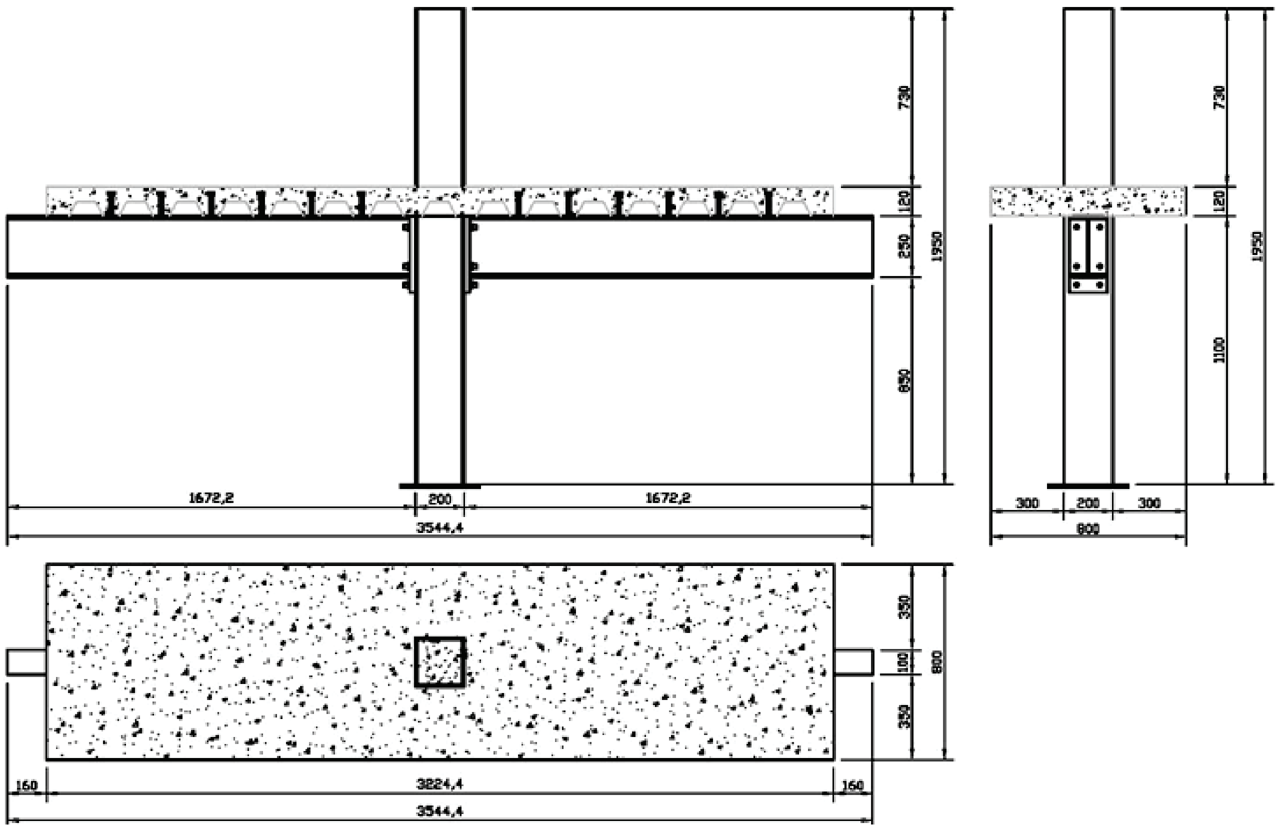


Figure 3 – Design of the slab reinforcement

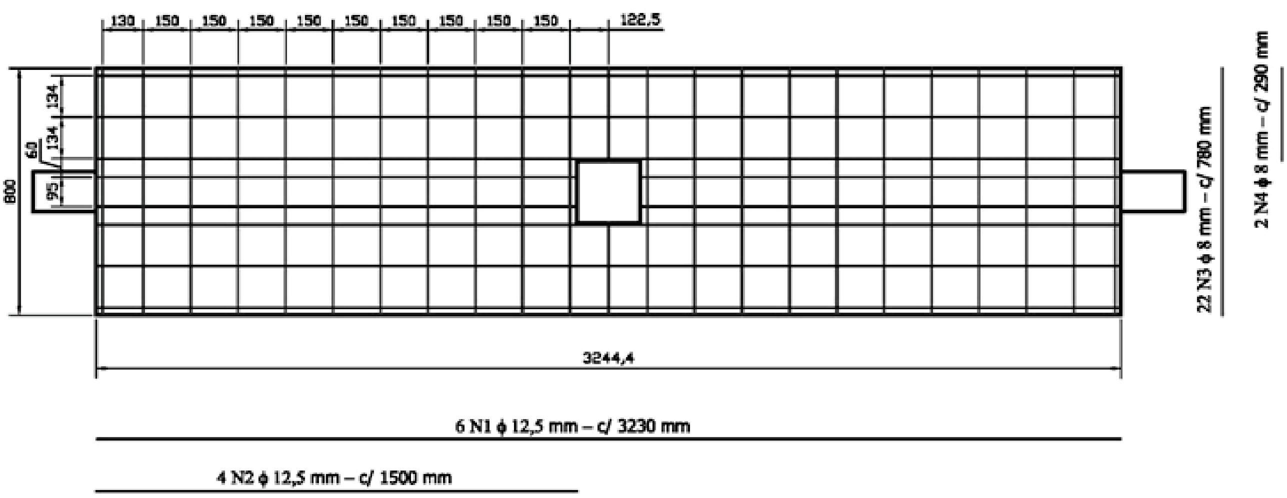
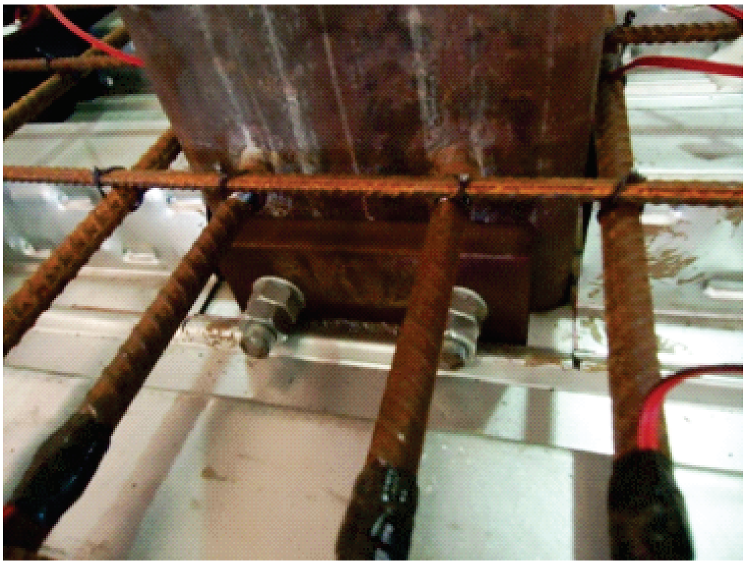


Figure 4 – Details of slab reinforcement



A Slab reinforcement

B Continuity bars linked to the column by gloves

to analyze the behaviour. The analyses were carried out based on the connection stiffness. The results of characterization tests were used to describe the properties of the elements in numerical model. At this paper, the connection tested is called Model 1 or Reference Model, therefore its behaviour was used as a base for numerical model calibration and parametric analysis.

Three hydraulic jacks were used for efforts application, each one with capacity of 500 kN. Two hydraulic jacks were located close to the end of the beams, distant from the connection 1580 mm, and the third one was used to apply a constant force on the top of the column in an attempt to cause the same effect of the efforts coming from the upper floors in a multistory building (Figure 6). This force was applied with a steel shape under the hydraulic jack that distributed the efforts by the column cross section top, with an intensity of 150 kN. The measurements of displacements and deformation were carried out with transducers which were positioned below the beams (Figure 7) and strain gauges glued in the steel bars and in the steel profiles, in order to identify when these elements reached the yielding stress.

Figure 5 – Scheme of cyclic loading application

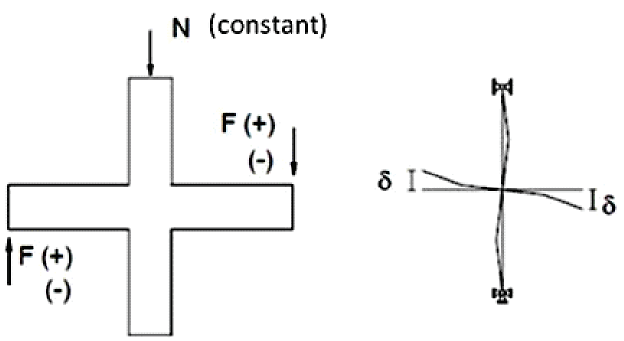


Figure 6 – Scheme of the test

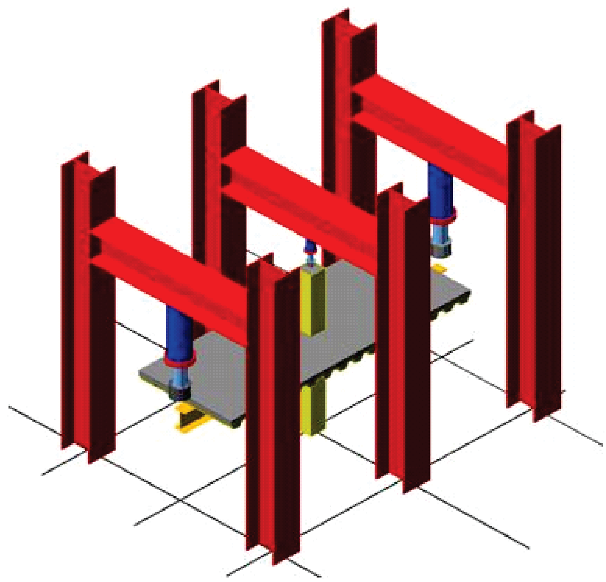


Figure 7 – Transducers position

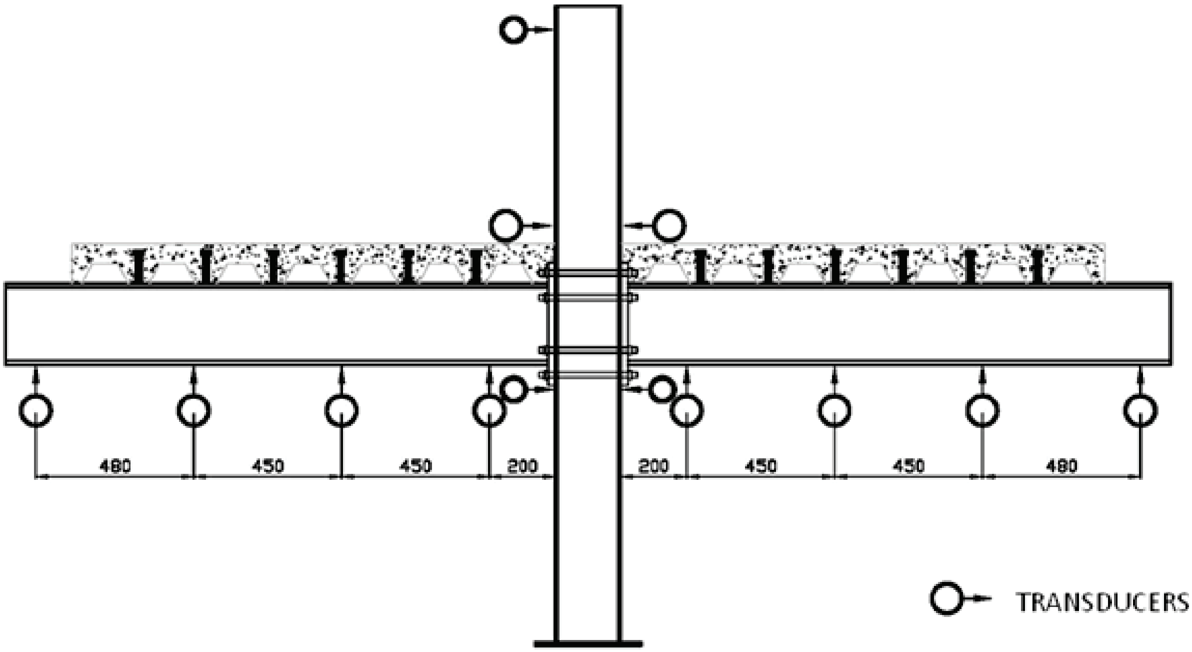


Figure 8 – Device used for force application in the beams

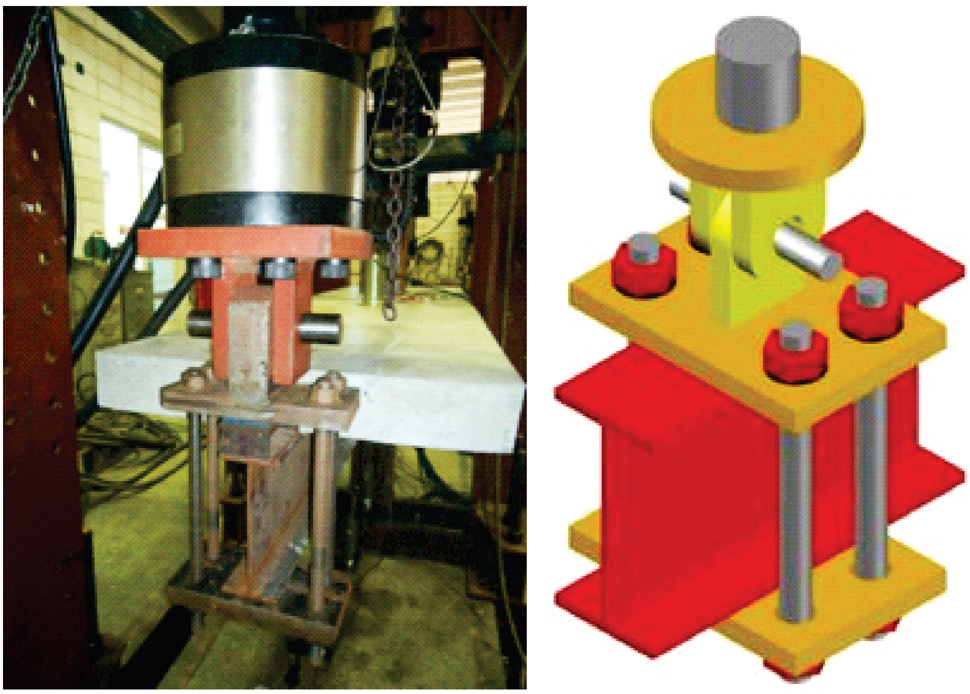
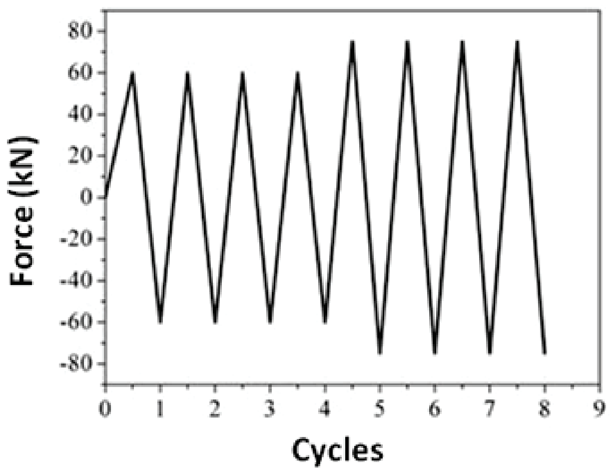


Figure 9 – Representation of loading cycles of Model 1



Based on measurements of displacement and applied force it was generated the bending moment curve *versus* rotation. Based on this curve it was possible to determine the stiffness of the connection. The measurement of displacement of the first transducer (closest to the connection, Figure 7) was used to determine the connection rotation. From these values it was calculated the arc tangent of the angle produced with the starting position of the beam. The arc tangent corresponds approximately to the connection rotation.

The cyclic loading was applied to the structure by a device developed specifically for this function. This device allowed the application of displacement reversals which generated in the connection the positive and negative bending moments at the same time. Another function of this device was to prevent the appearance of horizontal forces that could damage the hydraulic jack. Details of this device are shown in Figure 8.

The loading started with four cycles of 60 kN, then the loading was increased in 25%, reaching 76 kN, when four cycles more were performed (Figure 9). At the fourth cycle there was a failure of the structure with the detachment of the slab caused by the rupture of the concrete in the region of the shear connectors located near the point of force application, as shown in Figure 10. The intensity of the cycles was defined based on previous analytical studies conducted according to requirements code. It should be noted that another loading stage was provided with intensity equal to 95 kN, however, by reason of failure occurred in the slab-beam connection it cannot be achieved.

4. Experimental results

The maximum vertical displacements below the point of force application were 55.85 mm for the right side of the model and 54.46 mm for the left side to the force corresponding to 76 kN. The failure of the model occurred with the rupture of the connection between beam and slab, performed by shear connectors. Due to this occurred, it was not possible to reach the failure of beam-column connection, which according to the code [8] would occur when the hydraulic jack reached approximately 95 kN.

In Figure 11 is shown the force *versus* displacement curves wherein it is possible to note that there are not symmetric to the positive and negative bending moment. As a result of the asymmetric model, due to the presence of the slab, the bending moment *versus* rotation curves, presented in Figure 12, show that the bottom part of the connection rotated more than the top, for the both side of the model.

The stiffness for the last loading cycle of the left side of the model was 26398.87 kNm/rad for the bottom of the connection and 33656.42 kNm/rad for the top, respectively for positive and negative bending moment. For the right side, the difference be-

Figure 10 – Failure of the slab-beam connection

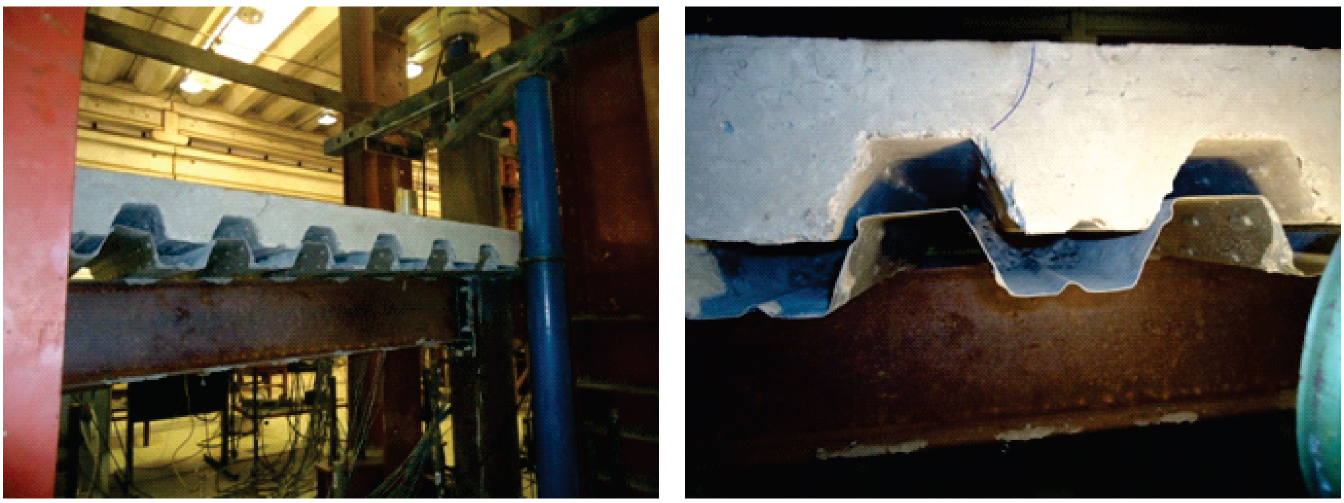
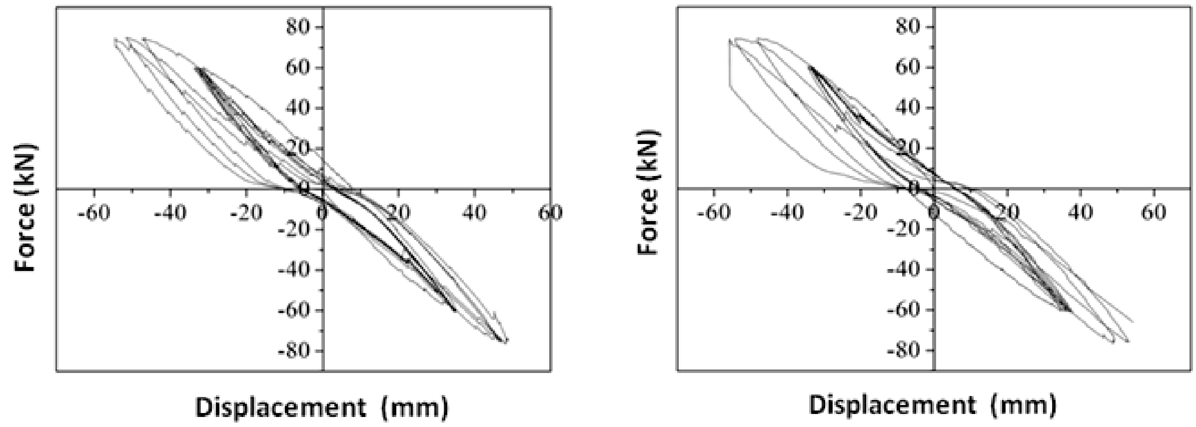


Figure 11 – Force *versus* displacement curves of Model 1



A Left side

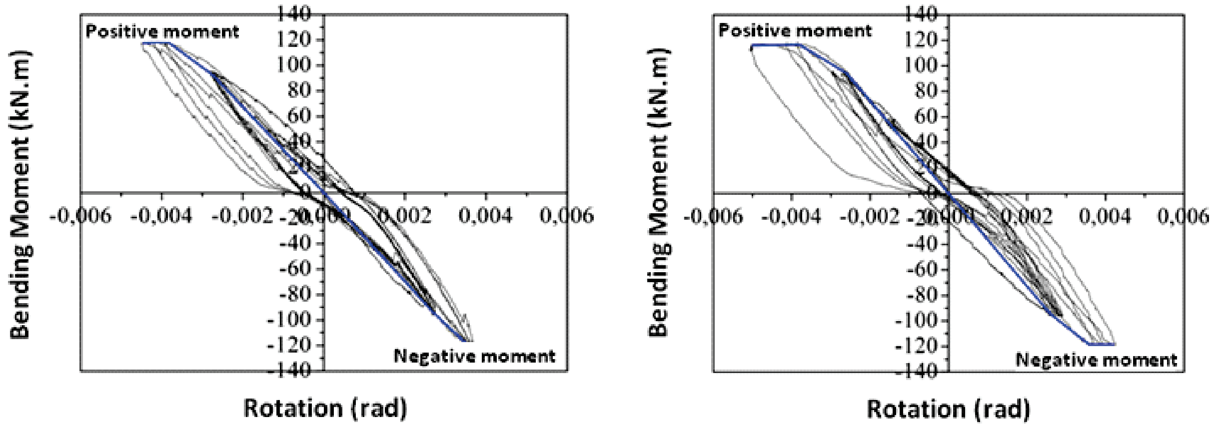
B Right side

tween the stiffnesses was lower than the left side because of the detachment of the slab has occurred on this side, with stiffness to the bottom of the connection equals to 28311.42 kNm/rad and 23178.00 kNm/rad to the top. Based on the envelope curve, the initial stiffness presented by the left side of the Model 1 was 34324.29 kNm/rad and the right side, 36360.76 kNm/rad, resulting in an average for the stiffness equal to 35342.52 kNm/rad. For this calculation it was determined the slope coefficient of the line through the origin, whose value corresponds to the stiffness of the connection.

It was observed in comparing the loss of stiffness that for positive bending moment, in both sides, the stiffness reduction was the same, decreasing at about 23%. For the negative bending moment, which is resisted by the slab reinforcement, there was some difference, justified by the fact that the detachment of the slab have occurred on the right side of the model. On the left side the loss of stiffness was only 2%, as shown in Table 1.

The connection which was examined experimentally was also classified according to their degree of stiffness to negative bending moment following the instructions of Eurocode 3 [9]. For this purpose some geometric properties of the beam had to be determined as the moment of inertia and modulus of plastic resistance and plasticization moment, which are shown in Table 2. Figure 13 shows the classification scheme of connections according to Euro-

Figure 12 – Bending moment *versus* Rotation curves and envelope curves of Model 1



A Left side

B Right side

Table 1 – Experimental stiffness of reference model

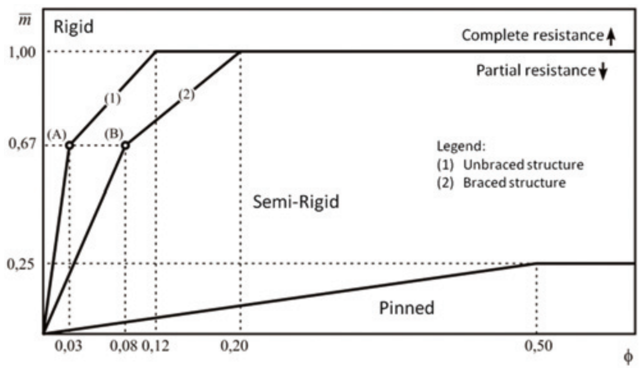
	Left Side			Right Side		
	Inicial	Final		Inicial	Final	
		Positive Moment	Negative Moment		Positive Moment	Negative Moment
Stiffness (k) kNm/rad	34324,29	26398,87	33656,42	36360,76	28311,42	23178,00
$k_{final}/k_{inicial}$	-	0,77	0,98	-	0,77	0,64

Table 2 – Beam properties used in the classification of the stiffness

Beam Properties	
Moment of inertia (Iz) cm^4	2887,30
Modulus of plastic resistance (Zx) cm^3	253,60
Plasticization Moment (Mp) kNm	88,76
Length (Lb) mm	1650,00
Yielding Stress (fy) MPa	350,00
Elastic Modulus (E) MPa	230000,00

code 3 [9] for unbraced and braced structures.
The classification was done for the situation that the connection was subject to negative bending moment, because it is the effort imposed to the structure for the most time of its life. The results obtained for both sides was that the slab contributed to resist the efforts, therefore the connection had the resistance enhanced and reached the classification of rigid with full resistance to braced structures, as can be seen in Figure 14.

Figure 13 – Connections classification specified by Eurocode 3



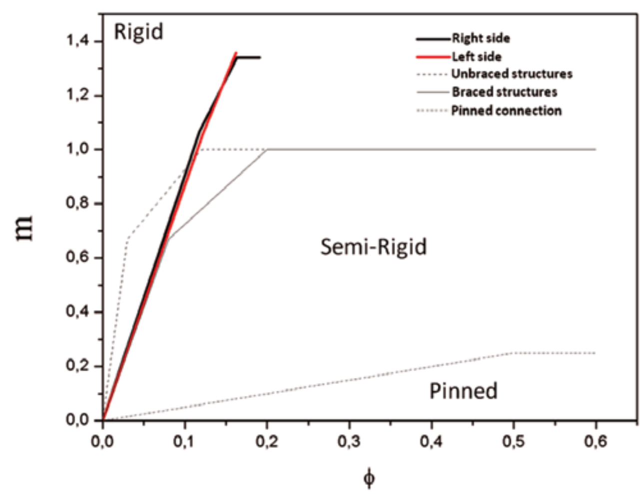
5. Numerical modeling

5.1 Materials

5.1.1 Properties

The material properties used in the numerical model were determined in the characterization tests, and others, such as fracture energy of concrete, which could not be determined experimentally, was determined based on requirements of the CEB MC 90 [10], assuming that in the production of concrete was used aggregates with maximum dimension of 19 mm. For the steel profiles the elastic modulus and yielding stress were determined by tensile tests according to ABNT NBR 6892:2002 [11] code.
The shear connectors used in the model were the same used in research [7], and the properties adopted for them were taken from there. For elastic modulus of the bolts and shear connectors were adopted nominal values characteristic of each material.
Related to the reinforcement bars of the slab, the properties adopt-

Figure 14 – Classification of model 1 by Eurocode 3



ed were the averages of values obtained from tensile tests. Table 3 summarizes the properties adopted for each element.

5.1.2 Constitutive Models

- **Concrete**
  The constitutive model used for the concrete was suitable for brittle or quasi-brittle materials (CONCRETE AND BRITTLE MATERIALS). To characterize the distribution of crack was used the TOTAL STRAIN model, whose the advantage is the simple concept. In the program DIANA the TOTAL STRAIN can be represented by ROTATING CRACK MODEL or FIXED CRACK MODEL. In the composite connection studied in this paper was used the FIXED CRACK MODEL. The tensile concrete behaviour was assumed as brittle and in compression was used an ideal elastic-plastic model.

- **Steel**
  Related to steel profiles, their constitutive model need to be describe by only two features: the yielding stress and hardening. The plasticity models of Tresca and von Mises are applicable to steel elements because they are ductile materials.
  The model of maximum energy distortion of Von Mises, chosen for the steel elements, admits that the maximum energy accumulated in the distortion of the material cannot be equal or greater than the maximum distortion energy for the same material in uniaxial tensile test.
  In summary, METAL model was adopted with the criteria of von Mises plasticity with IDEAL PLASTICITY, without consideration of the hardening or strain hardening. In the model of ideal plasticity, or also known as perfectly plastic, the material does not support efforts after to reached the yielding stress.

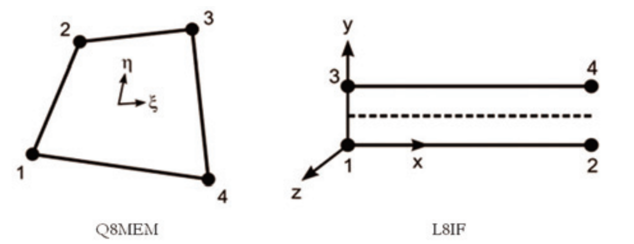
- **Interface**
  The DIANA has two families of interface elements: structural interface, for structural analysis, and structure-fluid interface, used for analysis of fluid and dynamic structure. There is another type of finite element that can be used in place of interface elements, which were the contact elements. These elements are usually used to analyze the contact between structural elements. The interface elements used in this study were structural interface.
  For the two joints considered in numerical models, between the column and endplate and between the beam and slab, the interface was represented by constitutive model for cracking

Table 3 – Material properties adopted in modeling

COLUMN CONCRETE CORE				
Tensile Strength (MPa)	Compressive Strength (MPa)	Elastic Modulus (MPa)	Fracture energy (Nm/m²)	Poison (υ)
3,59	54,81	38415,51	0,136	0,2
SLAB CONCRETE				
Tensile Strength (MPa)	Compressive Strength (MPa)	Elastic Modulus (MPa)	Fracture energy (Nm/m²)	Poison (υ)
3,77	51,41	34333,65	0,136	0,2

STEEL PROFILES		
Yielding Stress (MPa)	Elastic Modulus (MPa)	Poison (υ)
350,00	235000,00	0,3
BOLTS		
Yielding Stress (MPa)	Elastic Modulus (MPa)	Poison (υ)
350,00	200000,00	0,3
SHEAR CONNECTORS		
Yielding Stress (MPa)	Elastic Modulus (MPa)	Poison (υ)
250,00	200000,00	0,3
REINFORCEMENT		
Yielding Stress (MPa)	Elastic Modulus (MPa)	Poison(υ)
525,81 (f12,5mm)	201245,00	0,3

Figure 15 – Finit Elements (7)



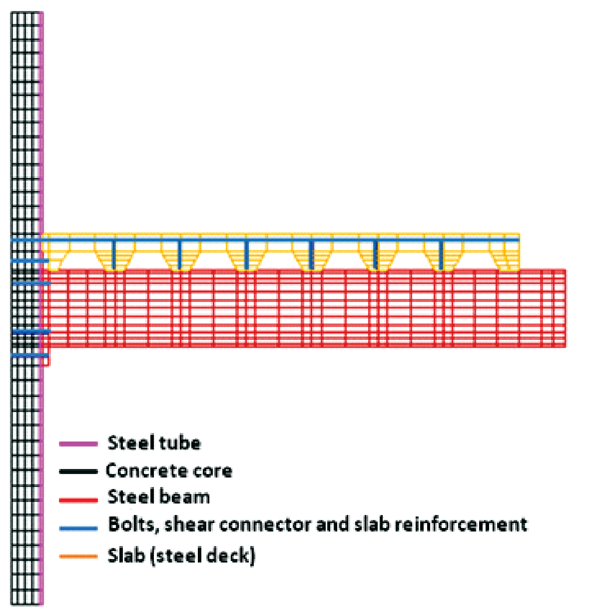
(CRACKING), with discrete cracking (DISCRETE CRACKING) and brittle behaviour.

- **Reinforcement, bolts and shear connectors**
  The bolts, the slab reinforcement and the shear connectors were represented by REINFORCE, a tool of the software DIANA specific to simulate the behaviour of steel bars. The finite element crossed by the REINFORCE is stiffened, causing the same effect that steel bars cause in reinforced concrete structures.
  To describe these elements was used the von Mises with ideal plasticity model, because they are steel elements, as already explained.

5.2 Finite element

Two types of finite elements were used to construct the mesh: elements of plane stress and interface elements.

Figure 16 – Finite elements mesh



The plane state elements were used to represent the concrete and steel, while the interface elements are used at the joint between the endplate and the column and between the beam flange and the slab. The finite element used for concrete and steel was the quadrilateral isoparametric element Q8MEM. This element has four nodes, two degrees of freedom per node, which represent the translations in x and y and linear interpolation function (Figure 15).
In the beam-column and beam-slab connections were used the L8IF interface element, which has 2 + 2 nodes with two degrees of freedom, relating to translations in x and y. This element is represented by two parallel lines in a plane configuration, as shown in Figure 15.
The structure of these elements describes the behaviour of the interface in terms of the relationship between the normal and shear forces at the site. According to [12], these elements are commonly used in the construction of meshes for numerical representation of masonry structures, connections and to describe the adherence along the reinforcement.

5.3 Mesh

The bidimensional model was selected to describe the connection because of this kind of modeling supplies the required information for analysis and provides less time in the processing of numerical models. Another thing used to decrease the time spent on modeling was the representation only the half of physical model.
The final finite element mesh presented 903 nodes and 775 elements. Figure 16 shows the finite element mesh with an indication of the structural elements that compose the numerical model.

Figure 17 – Boundary conditions of the numerical model

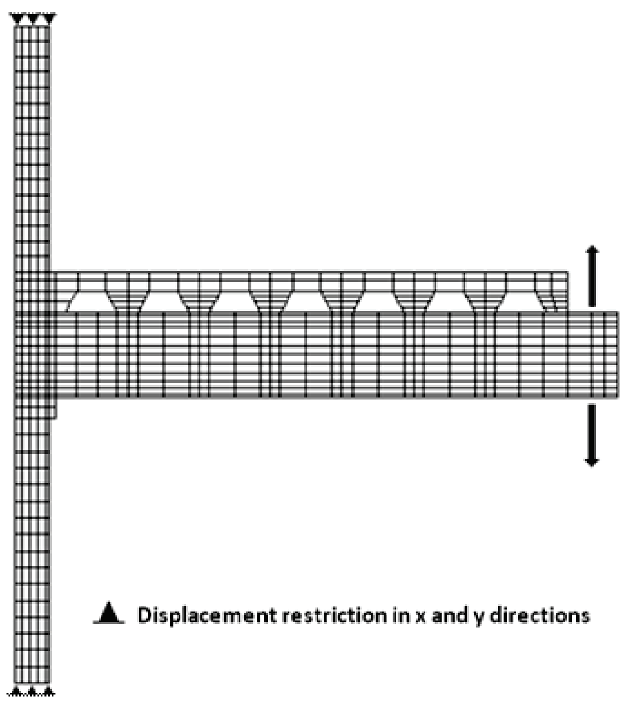
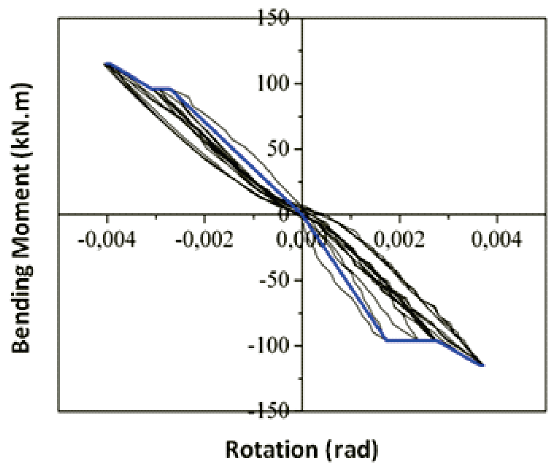


Figure 18 – Bending moment *versus* rotation curve of the Numerical Model



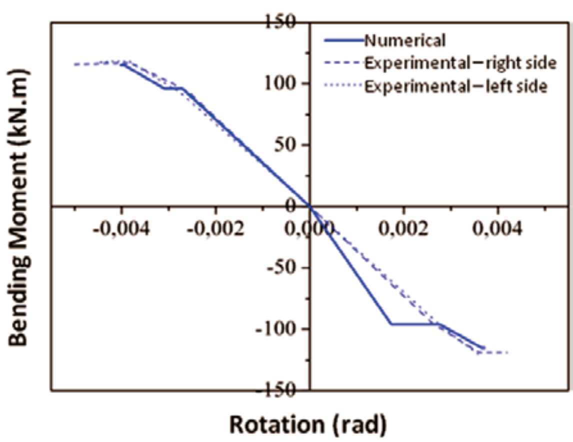
5.4 Boundary conditions

The boundary conditions adopted for the numerical simulation was the restrictions of displacements on x and y in the base and the top of the column to ensure the same test conditions. The application of loads was performed near the end of the beam, at 1580 mm from the connection. It was carried out force control. The representation of the boundary conditions is shown in Figure 17.

5.5 Nonlinear analysis

The nonlinear analysis was carried out in the numerical simulation with only consideration of physically nonlinear of the materials. To solve the nonlinear system was adopted the secant method with convergence criteria by rules of force and displacement.
Regarding the application of force, 557 load steps of 4 kN were performed, 15 load steps by half-cycle for amplitude equal to 60 kN and 19 load steps for amplitude of 76 kN. It was chosen this magnitude of steps to make the process faster, because there were no problems with convergence.

Figure 19 – Experimental and numerical envelope curves of the model 1



6. Numerical results

Figure 18 shows the bending moment *versus* displacement curve obtained with the numerical simulation with cyclic loading. According to this curve, to the negative bending moment the stiffness is higher, it is important to tell that in two directions of bending moment the maximum effort applied was 115.20 kNm. The maximum rotated for positive direction was 0.004 rad and for negative 0.0037 rad. In accordance with the envelope curve, the initial stiffness of the connection was 42552.23 kNm/rad.
Based on the results, a good correlation was observed between numerical and experimental envelope curves, mainly to the positive bending moment, as illustrated in Figure 19. The initial stiffness presented by the numerical envelope curve was 42552.23 kNm/rad and the average between the sides of the experimental model was 35342.52 kNm/rad. The numerical stiffness was 17% higher than the experimental stiffness.
The numerical stiffness was higher than the experimental stiffness because of the area of steel adopted in the simulation. It is difficult to know which exact area of steel is contributing to endure the efforts, thereby it was adopted the area corresponding to the four bars more closer to the column, totaling 490 mm² (4 f 12.5 mm). This criterion was adopted in compliance with the experimental results of deformation of steel bars. For the bolts was considered the area of two elements, equal to 400 mm ², for each row.

7. Parametric analysis

The parametric analysis was performed to identify important parameters in the design of the composite connection that would improve behaviour.

7.1 Diameter of the bolts

Related to the diameter of the bolts, three diameters were used in the parametric analysis, which were 12.5 mm, 20 mm and 25 mm. The other connection characteristics were maintained the same, as well as the properties of materials. The loading cycles applied in the simulations were the same used in the simulation of the reference model even as in the test.
Based on the reference model stiffness, which was 42552.23 kNm/rad, the model with bolts of 12.5 mm of diameter presented a stiffness decrease at about 31%. When the diameter was increased, in the case of 20 mm, 25% increase in diameter, the stiffness remained the same. In the other case, to a diameter of 25 mm, the stiffness increased 15%, as shown in Table 4. The representation of the bending moment *versus* rotation curves and the envelope curves are shown in Figure 20.

Table 4 – Relationship between the stiffnesses of the parametric analysis of the bolts diameter

Bolts diameter (mm)	Stiffness (kNm/rad)	$k_i/k_{\phi 16}$
16	42552,23	-
12,5	29385,91	0,69
20,0	42219,78	0,99
25,0	49017,47	1,15

7.2 Slab height

In reference connection model was used slab with 120 mm of height and to study the influence of this parameter was used height of 140 mm, 160 mm and 180 mm. It should be noted that in all of the models were used a concrete cover equal to 25 mm, like in the tested model, as specified in [13].
In Figure 17, the envelope curves from parametric analysis of the slab height compared with the envelope curve of the reference model are presented. As expected, all models with slab with higher height than the slab of reference model showed higher stiffness, but the differences were not significant. As shown in Table 5, for the slab with 140 mm of height, almost no difference in the stiffness was observed in comparison with the reference model, called Model 1 in Figure 21. For connection with slab with 160 mm of height the difference was 6% and for the highest slab, 180 mm, the difference was 8%. As can be seen also in Figure 21, the stiffness to the negative bending moment was higher in all cases, showing that the slab contributes with the increased of the stiffness. Another observation that can be made is that the height of the slab affects the connection stiffness in the same way for positive and negative bending moment. For positive and negative bending moment the tendency was equal for all heights.

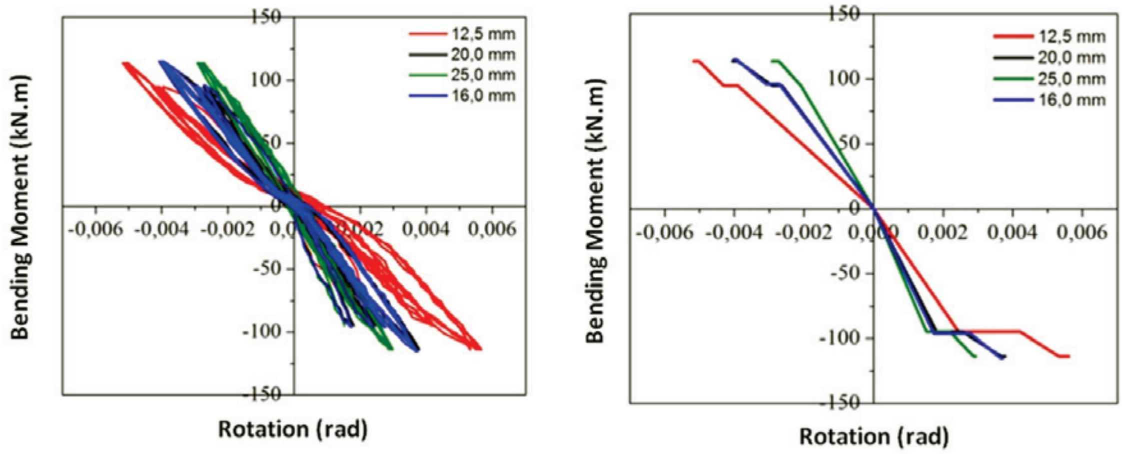
Table 5 – Relationship between the stiffnesses of the parametric analysis of the slab height

Slab height	Stiffness(kNm/rad)	$k_i/k_{reference}$
Reference L_{laje} = 120 mm	42552,23	-
L_{laje} = 140 mm	42171,03	0,99
L_{laje} = 160 mm	45381,65	1,06
L_{laje} = 180 mm	46042,11	1,08

7.3 Cross section of the beam

For the parametric analysis of the influence of the steel beam cross section on the behaviour of the connection were varied the thicknesses of the flanges and the web and the width of the flanges. These changes were made comprising three different cross sections: the first one with flange with 10 mm of thickness, the second one with flanges with width equal to 140 mm and in an attempt to increase the stiffness, the third cross section had flanges and web with thickness equal to 10 mm. The cross sections used in the parametric analysis are detailed herein Figure 22.
Comparing the bending moment *versus* rotation curves for all cross sections analyzed with the curve of the reference model (Figure 23), it was noted that the stiffness of all connections with robust beams was considered inferior and less rigid.
Examining the design process of the connection, the justification for the lower stiffness of the models of beam cross section parametric analysis was found. Therefore, as all other parameters of the connection were maintained, including the diameter of the bolts and their properties, and with the use of robust and less deformable beams, the bolts were more requested, becoming the weak point of connection, incompatible with the connected elements.

Figure 20 – Comparison of bending moment *versus* rotation curves and envelope curves used for parametric analysis of the bolts diameter



Bending moment *versus* rotation curves

Envelope curves

Table 6 shows the stiffness obtained in the simulation, including the comparison with the reference connection. The connection with beams with thicker flanges, the stiffness was 8% lower. However, the stiffness for the other two connections with cross section with wider flanges (section 2) and with thicker flanges and web (section 3), the difference was more significant, reaching approximately 12%. According to the results of the parametric analysis of the beam cross section was not possible to take an accurate understanding about the connection behaviour because of the diameter of the bolts was not suitable for the beams used. Nevertheless, the fact was important to attend the engineers to the correct connections design.

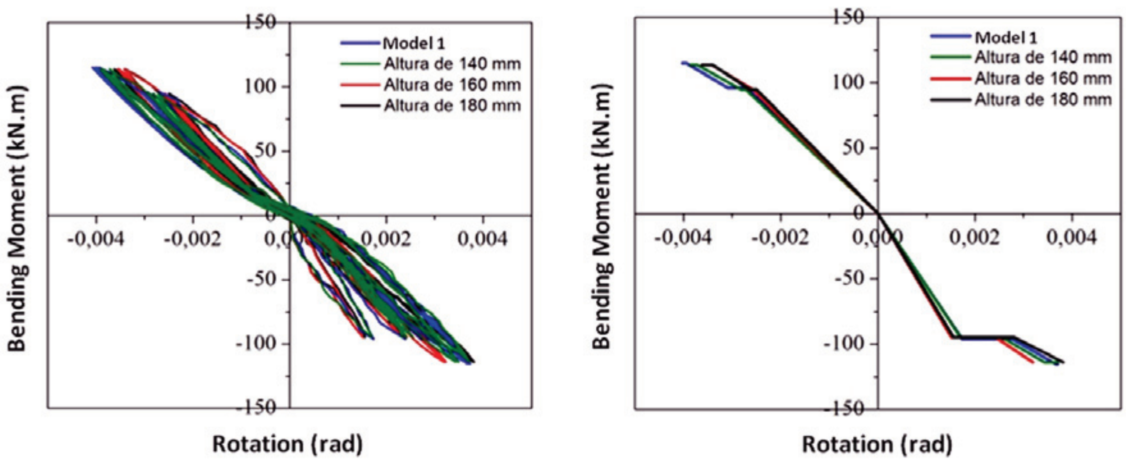
8. Conclusions

From the comparison between experimental and numerical results, it was concluded that the two-dimensional modeling portrayed satisfactorily the behaviour of the composite connection studied in this paper, providing advantages such as facility and quickness to create the model and reduced processing time.

Related to the parametric analysis, it was performed based on results of [14]. The diameter of the bolts, the beam cross section and the slab height were varied. For each parameter were adopted three variations and when one was changed, all others are maintained fixed to facilitate the comparisons. The main conclusions were:

- The use of bolts with larger diameters was the parameter that most influenced the behaviour of the connection. With the increases of the bolts diameter of 16 mm to 25 mm, at about 50% of increase, the stiffness increase 15%. The connection with bolts with 20 mm of diameter did not have change in the stiffness.
- The variation in the slab height also provided changes in the behaviour of the connection. When was used the height equal to 140 mm, the stiffness practically unchanged, but when the

Figure 21 – Comparison of bending moment *versus* rotation curves and envelope curves used for parametric analysis of the slab height



Bending moment *versus* rotation curves

Envelope curves

Figure 22 – Beams Cross sections used in the parametric analysis

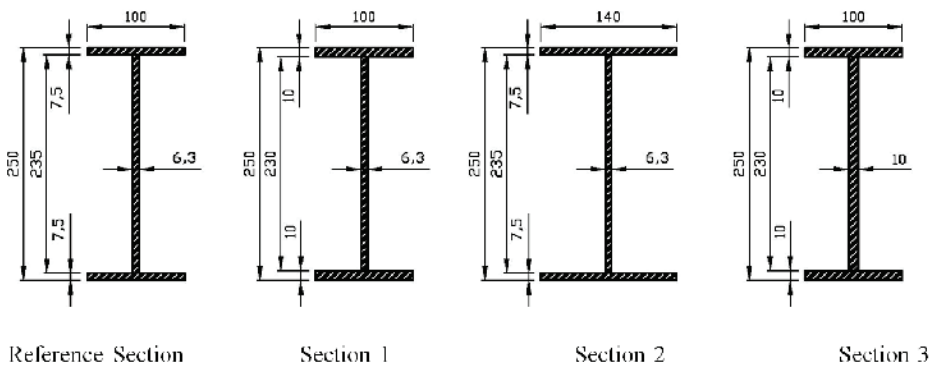
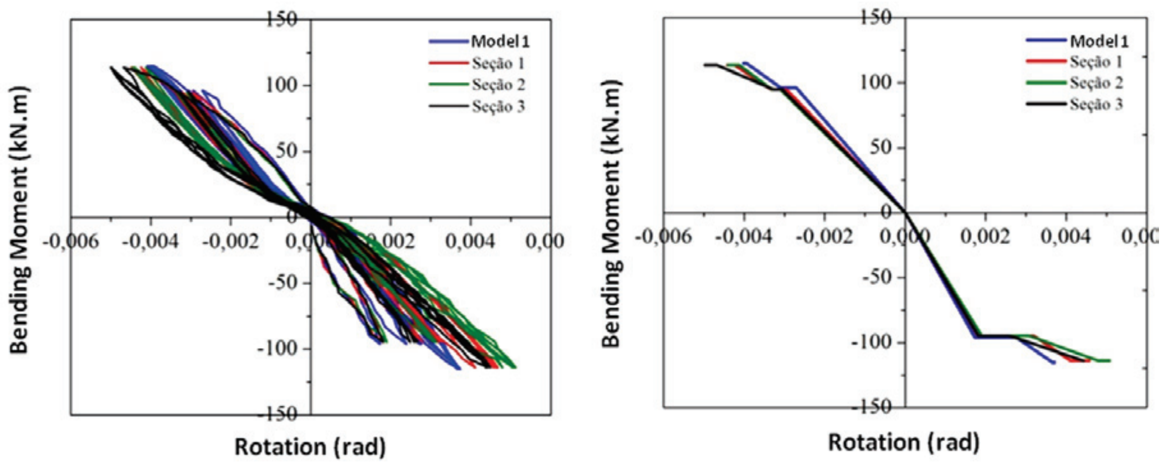


Figure 23 – Comparison of bending moment *versus* rotation curves and envelope curves used for parametric analysis of the beams cross section



Bending moment *versus* rotation curves — **Envelope curves**

height used was 160 mm there was an increase of 6%. When the height of the slab was increased to 180 mm, the stiffness increased at about 8% compared to the reference model. These data show that increasing the slab height, the stiffness of the connection will increase too, but there is a limit to the height that can provided significantly influence on stiffness.

- Based on the results of parametric analysis of the beam cross section was not possible to obtain any conclusion about the influence of this parameter in the behaviour of the connection. This fact occurred because of the bolts diameter (16.0 mm) was not suitable for robust beams, such as the sections 1, 2 and 3. After all, the event was important to alert the engineers about the correct connections design.

The final conclusion of the paper is that among the analyzed parameters the one with the most influence on the connection behaviour was the diameter of the bolts. Maintaining all other parameters equal to the reference model and increasing the bolts diameter in 50%, the stiffness was increased by 15%.

9. Acknowledgments

The authors express their thanks to the CNPq for a PhD scholarship, the FAPESP for the thematic project “Nucleation and growth of research, innovation and diffusion in precast concrete and composite structures for the construction of modernization”, which provided financial support for the construction of the experimental model, the Structures Laboratory of Department of Structural Engineering School of Engineering of Sao Carlos – USP which carried out the test as well as all persons who contributed to the testing.

Table 6 – Relationship between the stiffnesses of the parametric analysis of the beam cross section

Beam cross section	Stiffness (kNm/rad)	$k_i/k_{reference}$
Reference section	42552,23	-
Seção 1	39227,41	0,92
Seção 2	37157,68	0,88
Seção 3	38189,66	0,88

10. References

[01] TRAJANOSKA, B.; ARSOVA-MILOSHEVSKA, G.; BOGATINOSKI, Z. (2000). Numerical modeling of welded rigid beam-column connections at multi-storey structures. Balkanski. Sofia, Bulgaria.

[02] MAGGI, Y. I. (2004). Análise do comportamento estrutural de ligações parafusadas viga-pilar com chapa de topo estendida. Tese (Mestrado). Escola de Engenharia de São Carlos, Universidade de São Paulo. São Carlos. 269p.

[03] CHOI, S. M.; HONG, S. D.; KIM, Y. S. (2006). Modeling analytical momento-rotation curves of semi-rigid connections for CFT square columns and steel beams. Advances in Structural Engineering, Vol. 9, No. 5.

[04] HU, J. W.; LEON, R. T. (2010). Analyses and evaluations for composite-moment frames with SMA PR-CFT connections. Springer Science. v.65, p. 433 – 455.

[05] CONCEIÇÃO, J. L. (2011). Ligação mista viga-pilar resistente a momento. Dissertação (Mestrado). Escola de Engenharia, Universidade Federal de Minas Gerais. Belo Horizonte, MG. 155p.

[06] DE NARDIN, S. (1999). Estudo teórico-experimental de pilares mistos compostos por tubos de aço preenchidos com concreto de alta resistência.

Dissertação (Mestrado). Escola de Engenharia de São Carlos, Universidade de São Paulo. São Carlos. 148p.

[07] DE NARDIN, S. (2007). Investigação de dispositivos de ligação entre pilares preenchidos e vigas mistas em pavimentos mistos delgados. Relatório científico de Pós-doutorado. São Carlos. 149p

[08] ASSOCIAÇÃO BRASILEIRA DE NORMAS TÉCNICAS (2008). NBR 8800: Projeto de execução de estruturas de aço e de estruturas mistas de aço e concreto de edifícios. Rio de Janeiro – RJ.

[09] EUROPEAN COMMITTEE OF STANDARDIZATION (1993). Eurocode 3 – Design of steel structures. Part 1.8: Design of joints. Brussels.

[10] COMITÉ EURO-INTERNATIONAL DU BÉTON, CEB-FIP Model Code 1990 – Design Code, Thomas Telford Services Ltd., London, 1993, 437 p.

[11] ASSOCIAÇÃO BRASILEIRA DE NORMAS TÉCNICAS (2002). NBR 6892: Materiais metálicos – Ensaio de tração à temperatura ambiente. 34p. Rio de Janeiro – RJ.

[12] TNO BUILDING AND CONSTRUCTION RESEARCH, Diana User's Manual – Release 9, Delft, Netherlands, 2005, 622 p.

[13] ASSOCIAÇÃO BRASILEIRA DE NORMAS TÉCNICAS (2002). NBR 6118: Projeto de estruturas de concreto – Procedimento. Rio de Janeiro – RJ.

[14] KATAOKA, M. N. (2011). Estudo do comportamento de ligações viga-pilar preenchido submetidas a ações cíclicas. Tese de Doutorado. Escola de Engenharia de São Carlos, Universidade de São Paulo. São Carlos. 192p.