

A Study of Modelling Composite Connections in Two and Three Dimensions

M.N. Kataoka and A.L.H.C. El Debs
Department of Structural Engineering
University of São Paulo, São Carlos, Brazil

Abstract

In recent years, concrete filled steel tube (CFT) columns have become widely accepted and used in multistorey buildings as well as bridges. These elements provide ductility and toughness associated with the steel structures and high compressive strength associated with confined concrete. These advantages prevent the local buckling of the steel tube and they are especially efficient in moment-resisting frames in high seismic areas. The bolted endplate connection studied in this paper was realized between a CFT column and a steel beam. The combination of simple fabrication and speedy site erection have made the bolted endplates one of the most popular methods of connecting members in structural steelwork frames. In the present study, a nonlinear finite element analysis of a composite connection is presented. The nonlinear analysis results in a better understanding of the mechanical behaviour of a structure during its loading process until failure. Two numerical models were performed to describe the composite connection, a two and a three dimensions. The numerical models were calibrated with experimental data from full scale test. The objective of this study is to demonstrate that in some cases a simple numerical model is enough to evaluate some aspects of the connection as the force *versus* displacement behaviour. To obtain the correct stresses in the steel elements and in the bolts, it is recommended to use a refined model.

Keywords: non linear analyses, composite connection, concrete filled steel tube.

1 Introduction

The concrete-filled steel tube (CFT) is acknowledged as a very economic and effective structural system. This structural element has been studied by many researchers and has been widely used in the field because of the high ductility and large energy absorption capacity.

The results presented in this paper are part of a comprehensive research about steel-concrete composite connections involving CFT column that was conducted to understand their mechanical behaviour and provide design provisions. From the wide range of ductile connections configurations available, the focus of this paper is on the endplate type which is widely used in seismic resistant frames. In order to investigate the behavior of the proposed connection, numerical simulations with three-dimensional and two-dimensional models were performed.

The aim of this investigation is to employ the nonlinear finite element program DIANA to perform numerical simulations of the endplate bolted connections subjected to cyclic loading and compare the results to the experimental data. The experimental results used in this paper are from Kataoka [1] that tested some types of composite connections involving CFT columns.

The numerical analysis has been increasingly used to study the behavior of structures with some degree of nonlinearity, which was not possible before because of the absence of computing resources and physical models that can represent adequately the materials behavior. In [2, 3] were carried out numerical simulations of such a connection in precast concrete structure using two-dimensional model. The results portray satisfactorily the behavior of the structure with the application of both loading: monotonic and cyclic. As an example of more refined modeling has [4] which created a three-dimensional model of a bolted connection with a square cross section concrete-filled steel tube as a column, whose program ABAQUS was used.

2 Experimental Study

2.1 Experimental program

An experimental research was carried out on a connection between steel beam and CFT column by endplate and bolts. The test arrangement is basically a cantilever type and the model was subjected to cyclic loading in altering directions, as shown in Figure 1.

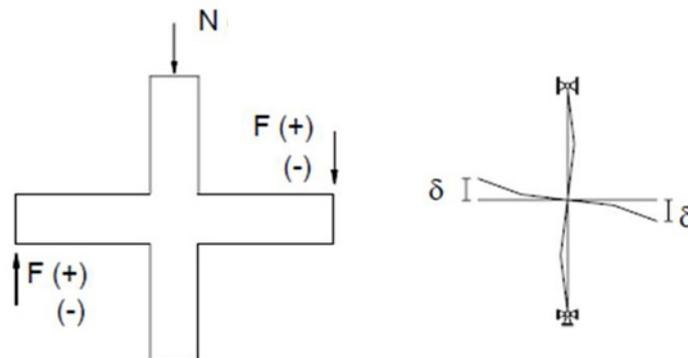


Figure 1: Scheme of application of cyclic loading.

The connection was realised between a concrete-filled steel tube 250 x 250 x 8 mm in size and two steel beams H250 x 100 x 6.3 x 7.5 mm made with ASTM A-36 steel. It was used a endplate with 22.2 mm of tickness and bolts with 16 mm of diameter. The eighth bolts passed through the steel tube and was adherent to the concrete fill. Design details and total dimensions of the physical model are shown in Figure 2 and 3.

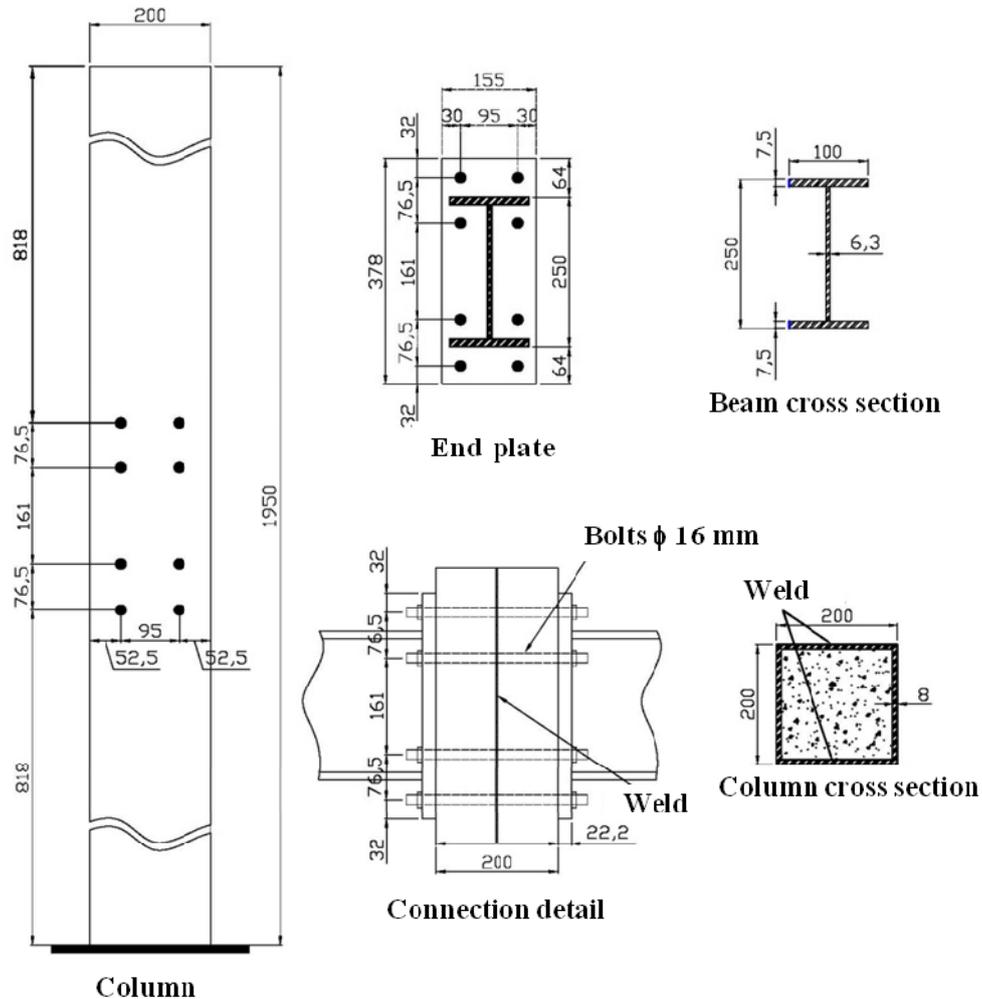


Figure 2: Design of the connetion.

To analyze the behavior of the connection were made measurements of the displacement under the beam, loading applied and the strain of the steel elements. In the test setup was used three hydraulic actuators, two of them in the tip of the beams and another one in the top of the column, in order to stabilize the model. The base and top of the column had their lateral displacement prevented by steel profiles with the also function of strengthening the stability of the prototype. The hydraulic actuators at each beam tip applied the reversible cyclic loading with loading control, as shown in Figure 4.

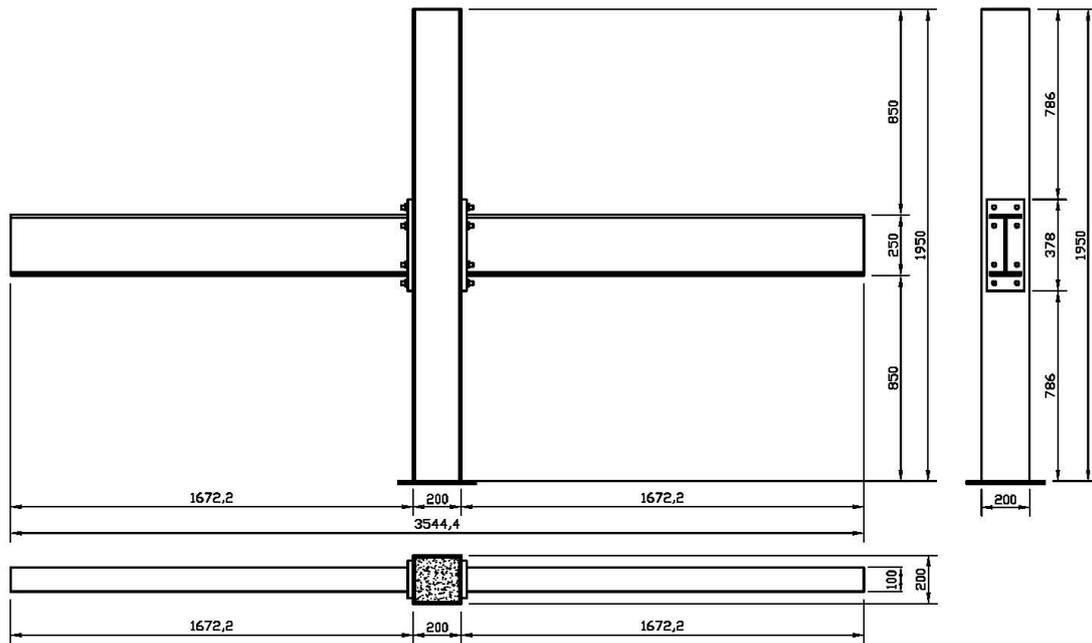
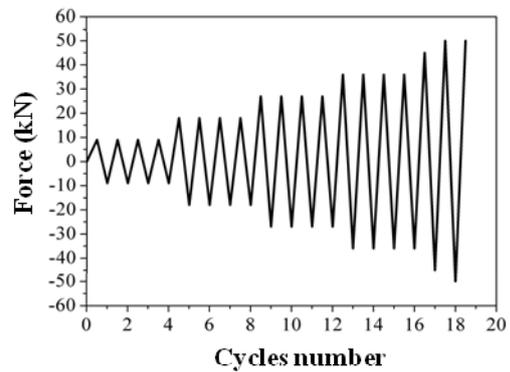


Figure 3: Dimensions of the physical model.



(a) Test setup.



(b) Cyclic loading.

Figure 4: Experimental test.

2.2 Test results

The cyclic loading applied to the prototype was divided into four stages of loading and after that the failure. In each stage of loading were conducted four cycles the same intensity. The first stage reached the force of 9 kN, 18 kN the second, third and fourth stage 27 kN 36 kN. After this last phase, the force was increased until the failure, which occurred with 50 kN and 18 cycles performed.

The maximum force achieved in the study was 50 kN, as shown in the force *versus* displacement curves in Figure 5. The maximum vertical displacements obtained for the ultimate level, measured under the points of loading application were 20.93 mm for the right side and 29.74 mm for the left side.

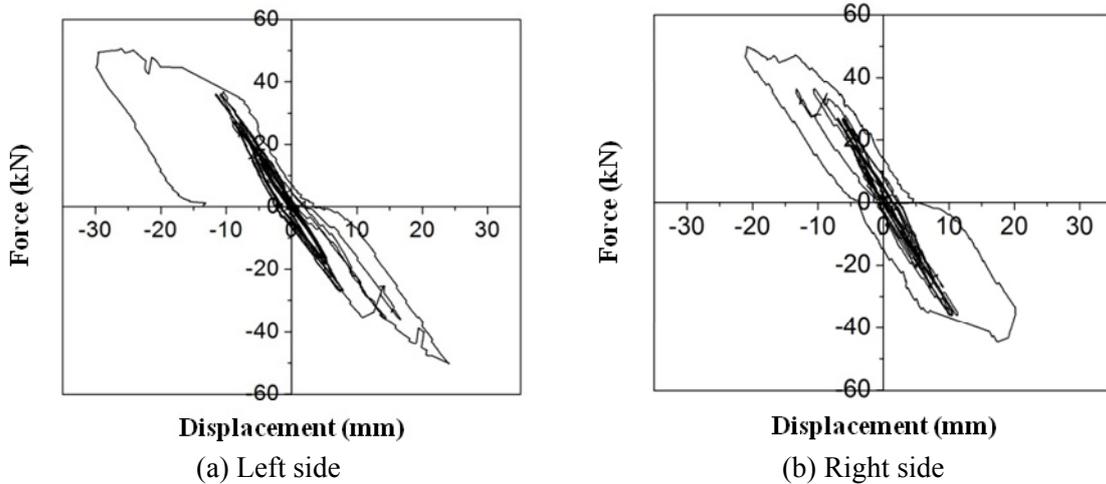


Figure 5: Hysteretic curve, force *versus* displacement, of the both sides of the model.

As the bolts strain was not measured during the test, some calculations were realized to obtain this information. In the Figure 6 is the scheme that explains the procedure adopted. The yielding stress determine to the bolts was 413.73 MPa, obtained by the bending moment generated in connection. In tests performed to characterize the bolts material, the yielding stress obtained was 356.25 MPa, lower than the value obtained in the connection test.

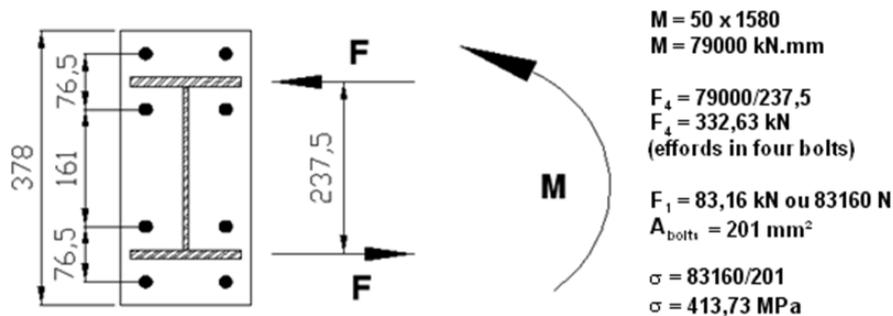


Figure 6: Scheme to determine the bolts strain.

3 Numerical Study

To better understanding the behaviour of the structure during its loading, it was used a nonlinear analysis in the numerical simulations. In the present study, the models were analysed using the DIANA software to conduct the study the composite connection.

According to some researchers, the use of modeling in structural analysis is an economical and practical way to predict the behavior of structures in engineering. An example is in Soriano [5] that claims that these analyses have motivated a great development of mathematical models, allowing realistic modeling, reliable and practical applications in engineering much more economical than physical models.

3.1 2D finite element model

3.1.1 Specimen modeling

The two-dimensional numerical model, as in the experimental model, consists of a column, a beam with the endplate and the bolts. Because it is a two-dimensional modeling were given only four bolts, each one represented by the cross section area of two units, as shown in Figure 7. To simplify the framing of the model only the half of this was modeling, with the symmetry in the center of the column, as shown in Figure 7.

Some simplifications were made in this model as the consideration of perfect adhesion between the concrete core and the steel tube. Another device used to construct the simple numerical model was the use of an interface element (L8IF – two + two nodes) in the face contact between the tube and the endplate. To this interface element did not resist any effort, its stiffness was close to zero, so that, the bolts were the resistant element in the connection. The bolts were discretized using a feature of the program called Reinforce, specific to reinforcement.

The finite element mesh was generated with 682 elements and 771 nodes and is presented in Figure 7. The finite element used for concrete and steel was the isoparametric with four nodes Q8MEM compatible with the interface element (L8IF). Details of the finite elements used in modeling are listed in [6].

The concrete was considered isotropic in elastic linear range and orthotropic in non-linear range, showing elastic behavior in compression and brittle in tension. The steel elements were considered as isotropic material in elastic linear range with Von Mises yield surface and ideal plasticity defined by the yielding stress of the material, taken at 350 MPa in non-linear regime.

Based on the scheme used in the test of physical model, the boundary conditions adopted in the numerical simulation were the displacement restrictions in the x and y directions applied at the base and top of the column. The analysis was performed with loading control, with the effort being applied to a point near the tip of the beam, located at 1580 mm from the connection, as shown in Figure 7. The materials properties used in the modeling are shown in Table 1.

Properties	Steel	Concrete	Bolts
Elastic Modulus (N/mm ²)	235000.000	40606.000	200000.000
Poisson's Ratio (ν)	0.300	0.200	0.3000
Fracture Energy (N/mm)	-	0.136	-
Tensile Strength (N/mm ²)	-	3.270	-
Compressive Strength (N/mm ²)	-	56.380	-
Yielding Stress (N/mm ²)	350.000	-	350.000

Table 1: The materials properties of two-dimensional model.

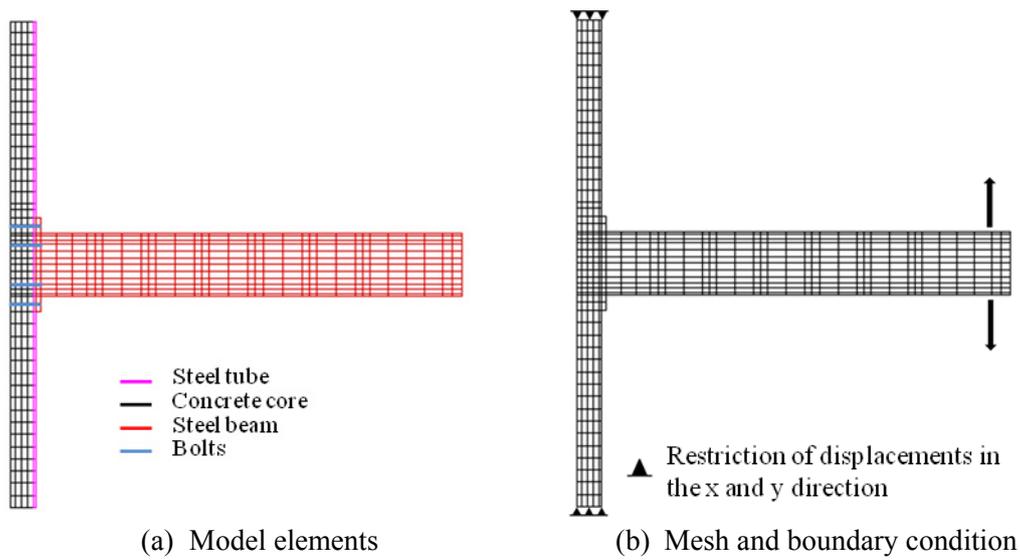


Figure 7: Mesh and boundary constrain of two-dimensional model.

3.1.2 Results

The loading cycles applied in the numerical model were the same cycles applied in the experimental model and, as shown in Figure 8, for the cycles with low intensity of loading, the curves overlapped, just presenting less stiffness to loading values greater than 35 kN. To the maximum loading applied in the model (50 kN) the displacement in local of effort application was approximately of 20 mm.

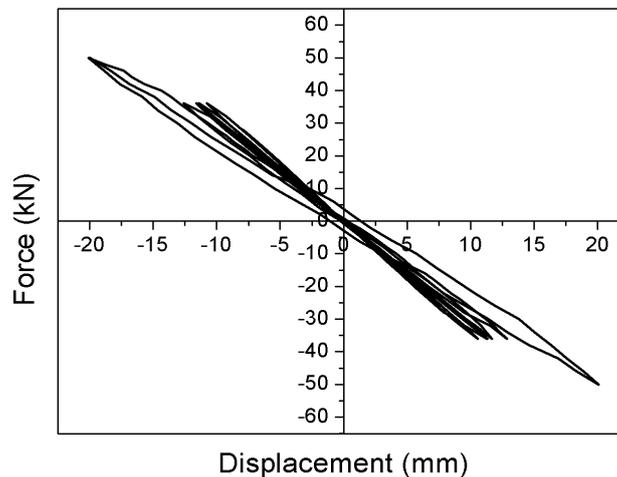
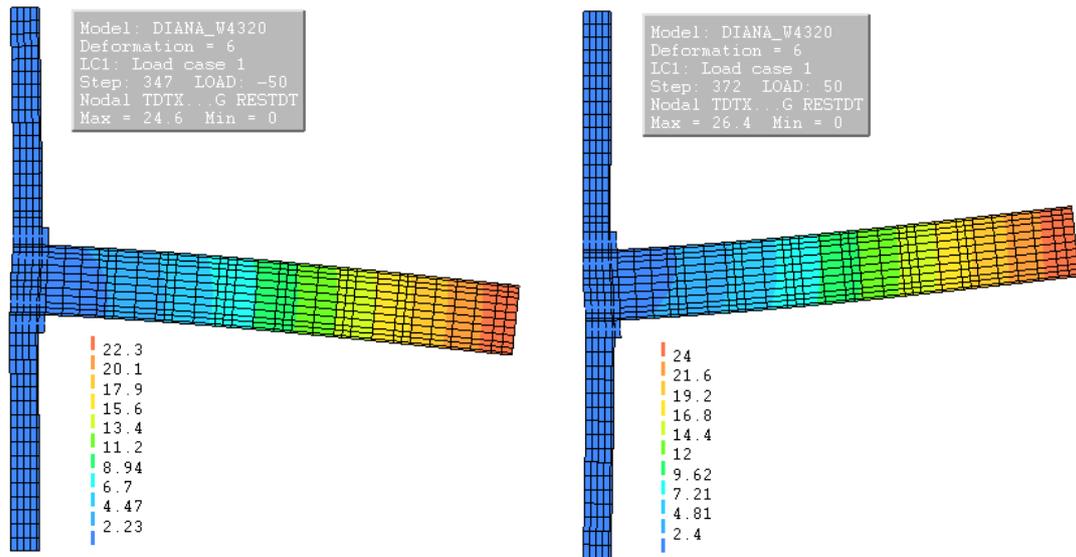


Figure 8: Hysteresis loading *versus* displacement of the two-dimensional model.

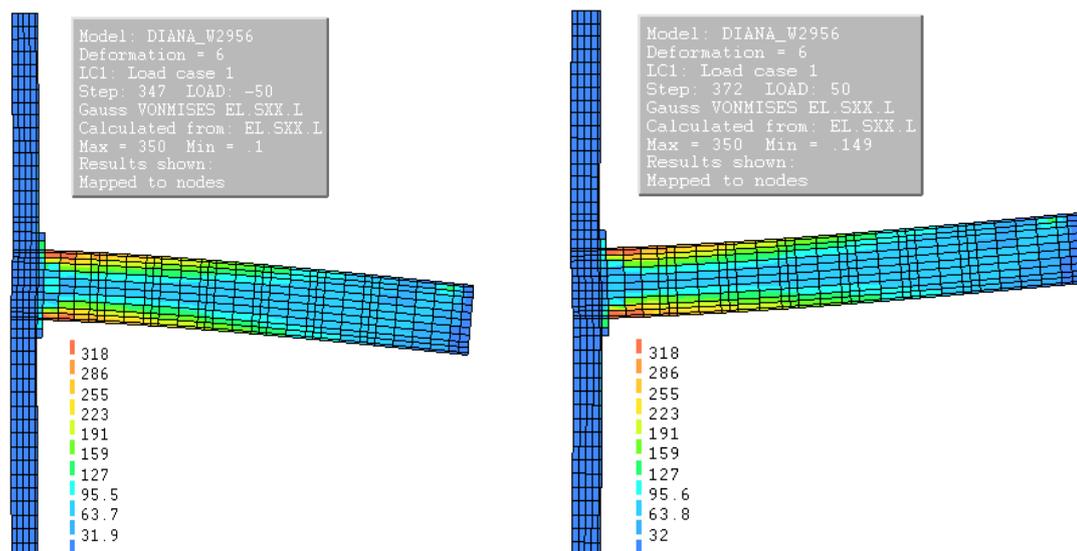
The Figure 9 illustrates the distribution of the displacements throughout the two-dimensional model at the peak of the last loading cycle. According to the results of numerical simulation, the maximum displacement at the end of the beam was 24.6 mm for negative bending moment and 26.4 mm to positive.



(a) Displacement in negative bending moment (b) Displacement in positive bending moment

Figure 9: Representation of displacements in the two-dimensional model.

For the last stage of cycles, the beam and the bolts were already at the stresses beyond the yielding stress. The beam had Von Mises stresses in the order of 350 MPa in the regions upper and lower of the flanges, stress like the yielding stress adopted for the material, according to the characterization tests (Figure 10). The bolts also reached the yielding stress, 350 MPa. These stress values were observed for both positive and negative bending moment. In Figure 11 is shown the distributions of stresses along the beam and the bolts.



(a) Von Mises Stresses in negative bending moment (b) Von Mises Stresses in positive bending moment

Figure 10: Representation the stresses in complete two-dimensional model.

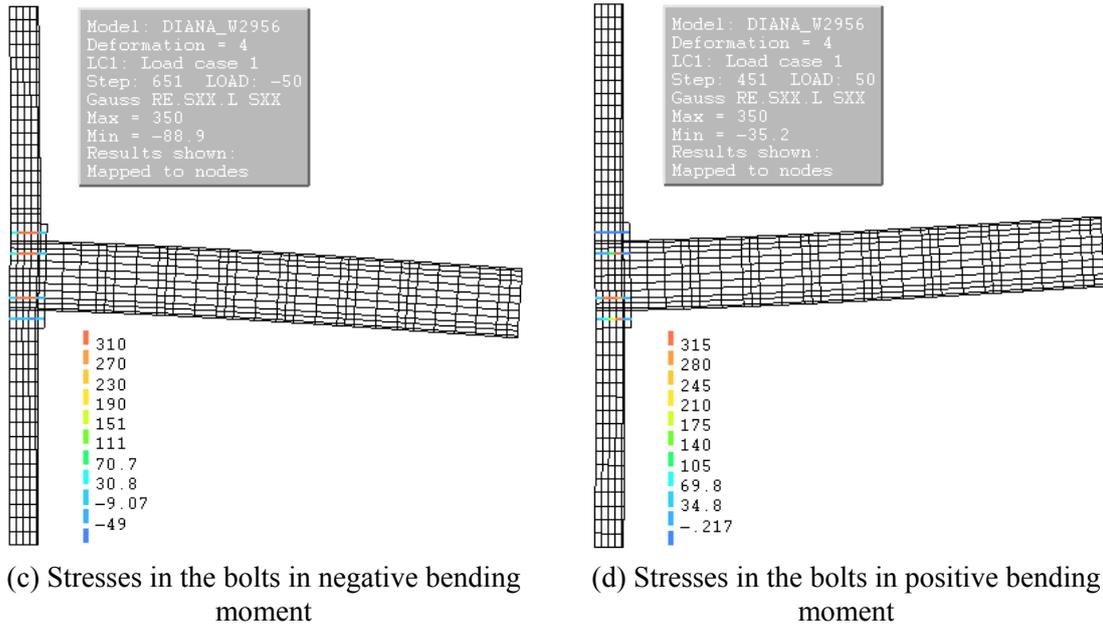


Figure 11: Representation of the bolts stresses in two-dimensional model.

3.2 3D finite element model

3.2.1 Specimen modeling

The geometry of the refined model was created using Midas FX+ 9.2.4 and for the processing was used DIANA 9.2.4. The software Midas FX + was chosen because it has simple commands to construct complex models. As shown in Figure 12, it replicates a composite connection from the experimental program of Kataoka [1].

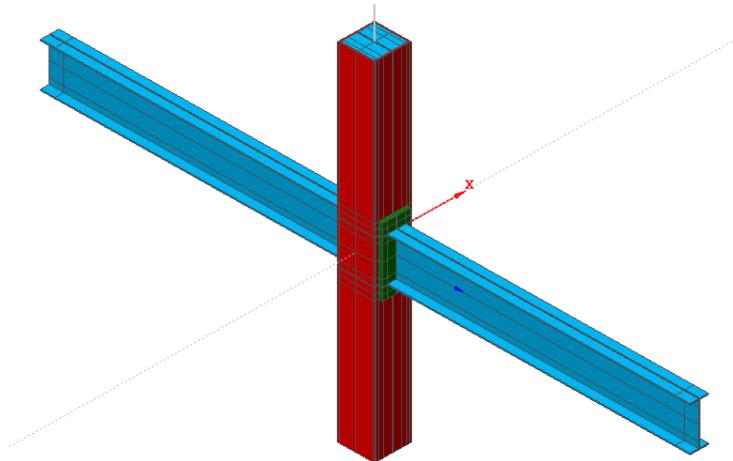


Figure 12: Refined three-dimensional model.

As shown in Figure 12, in the refined model the entire structure was modelling. It was made to obtain behaviour more similar to the physical model. The boundary condition and method of loading adopted in the finite element analysis followed closely those used in the tests and the two-dimensional model.

They were restricted the displacements in the x, y and z directions at the base of the column and in the top just in the directions x and y the displacements were restricted. In the top of the column was applied a displacement of 0.25 mm before the test begin.

Different mesh sizes have been examined as well to determine a reasonable mesh that provides both accurate results with less computational time. The mesh selected was generated with 5304 elements and 7235 nodes and is shown in Figure 13.

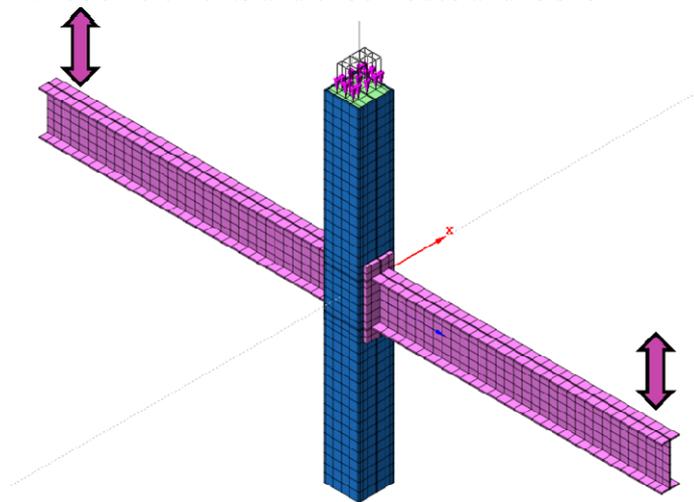


Figure 13: Mesh of three-dimensional model.

The solid finite element used to describe the concrete and the steel elements was HX24L that has eight nodes. To the interfaces (core concrete- tube and beam-tube) was selected the interface element Q24IF that has 4 + 4 nodes and it is used between two planes in a three-dimensional configuration. The bolts were described as a Reinforce, like in the two-dimensional model, with a cross section area of 200 mm² (16 mm of diameter).

The Mohr Coulomb yield surface was adopted to the concrete and to the steel elements was used the Von Mises yield surface with yield strength equal to 350 MPa to the beams and 272 MPa to the steel tube, as determined by characterization tests. The material properties of three-dimensional model are shown in Table 2.

Properties	Steel	Concrete	Bolts
Elastic Modulus (N/mm ²)	235000.000	40606.000	200000.000
Poisson's Ratio (ν)	0.300	0.200	0.3000
Cohesion (N/mm ²)	-	3.000	-
Friction Angle (Degree)	-	36.000	-
Yielding Stress (N/mm ²)	350.000	-	350.000

Table 2: The materials properties of three-dimensional model.

3.2.2 Results

In the case of refined connection model, the loading cycles were applied like in the test and in the two-dimensional model. The curve force *versus* displacement of the three-dimensional model (Figure 14) presented a little bit difference from the curve obtained to plane model. The stiffness of the refined model did not reduce with the loading cycles, fact that happened with the plane model. It can be explained by the distortion in the region of the connection provided by the behavior of the complete model (without symmetry). The vertical displacement of the beam was 18 mm, which was measured under the position of the loading. The maximum loading value was 50 kN.

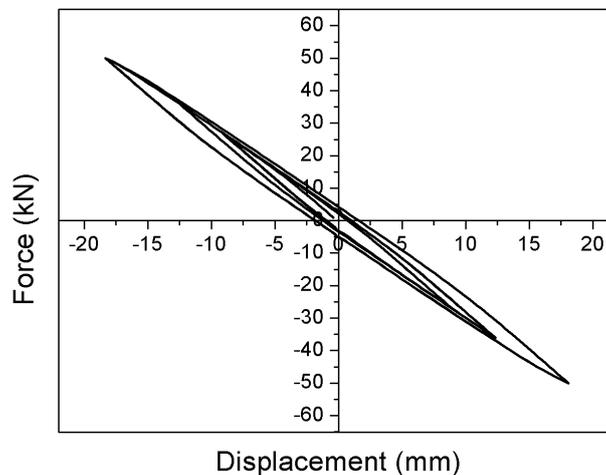


Figure 14: Hysteresis loading *versus* displacement of the three-dimensional model.

In Figure 15 is presented the configuration of the vertical displacement for the last cycle of 50 kN in the positive and negative bending moment. In the plane figure it is possible to note the distortion or rotation of the column in the connection region. This phenomenon involves many other changes in the behavior of the connection compared to simulations of the plan model such as the stiffness reducing and strain relieving in the bolts. In this same figure can be noted that in the end of the beams occurs the maximum vertical displacement (red region), local where the efforts were applied.

According to the results of the simulation, the failure of the model occurred in the beams, which exceeded the yielding stress. In Figure 16 is showed the stresses in the model in the final of the simulation. The distribution of stresses shows that the beams flanges, near de endplate, and in the face of the tube were the locals that developed highest stresses, exceeding the yielding stress. In these regions the stress reached 389 MPa (yellow region).

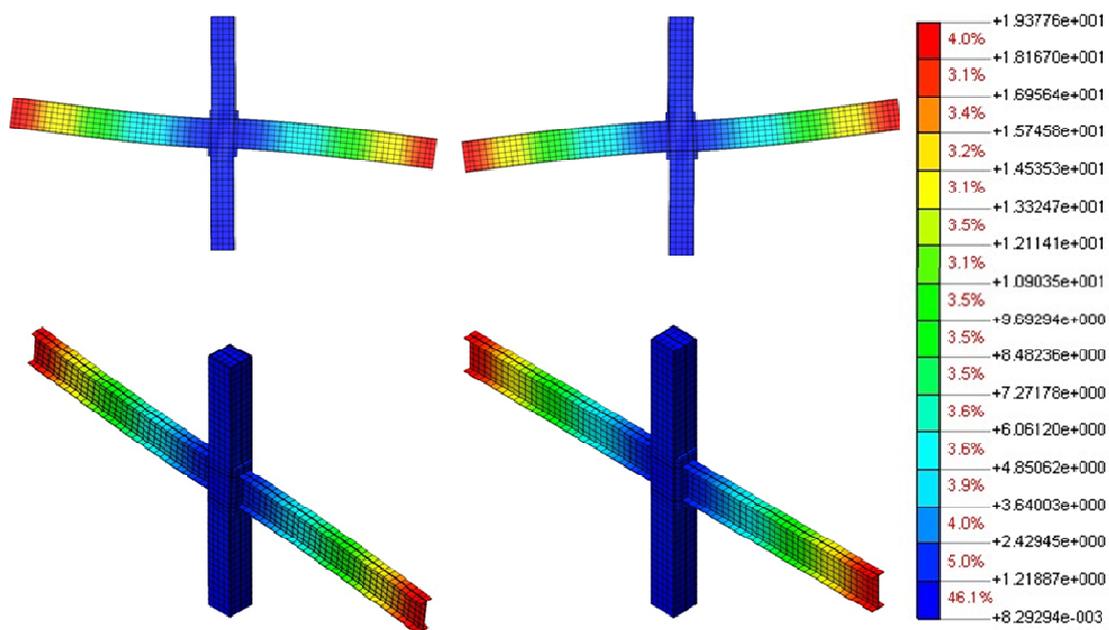


Figure 15: Representation of displacements in the three-dimensional model.

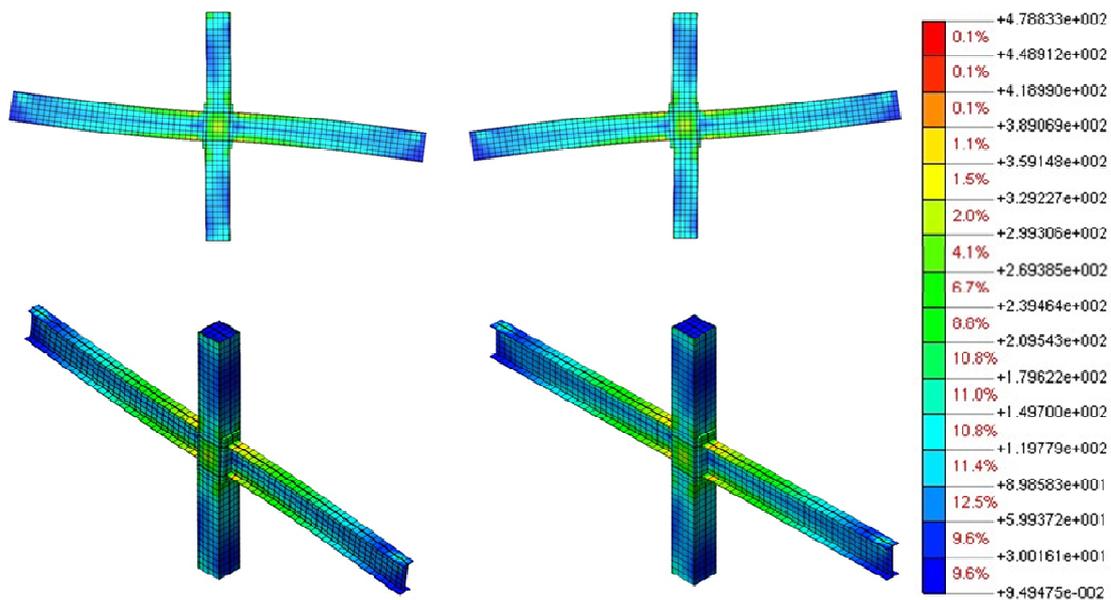


Figure 16: Representation of the Von Mises stresses in the three-dimensional model.

The stresses in the bolts were lower than the stresses provided by the two-dimensional modeling. The yielding stress of the bolts is 350 MPa, according to the material property, and in the simulation of the three-dimensional model, just 10% of this value were requested, at about 31 MPa.

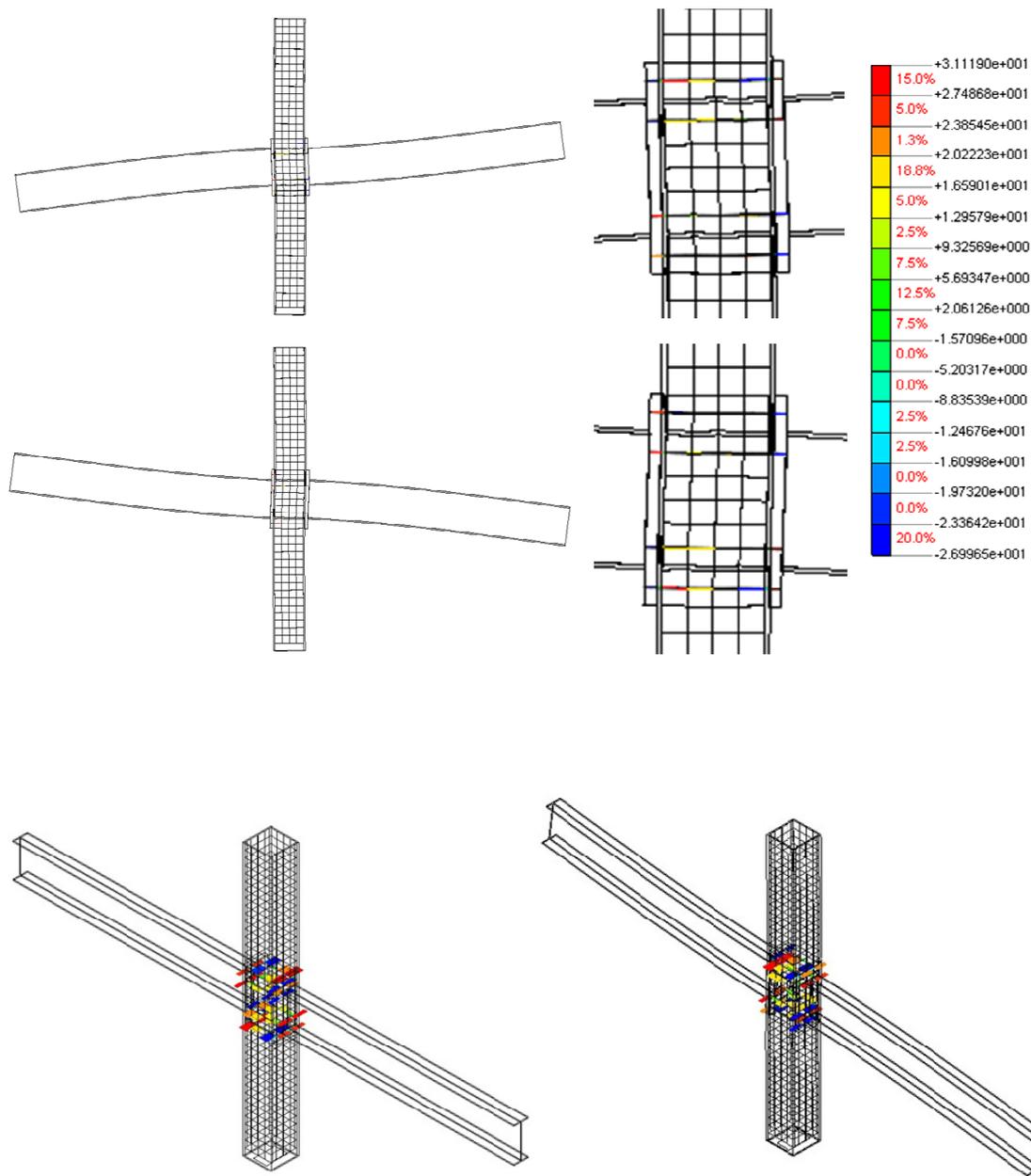


Figure 17: Representation of the bolts stresses in three-dimensional model.

As previously mentioned, the behavior of the three-dimensional model presented some differences from the plane model because of the detail included by the fact that it has been discretized the entire model. In the Figure 18 is showed the deformation of the CFT column that caused the stress relieving of the bolts and the decreased of the connection initial stiffness.

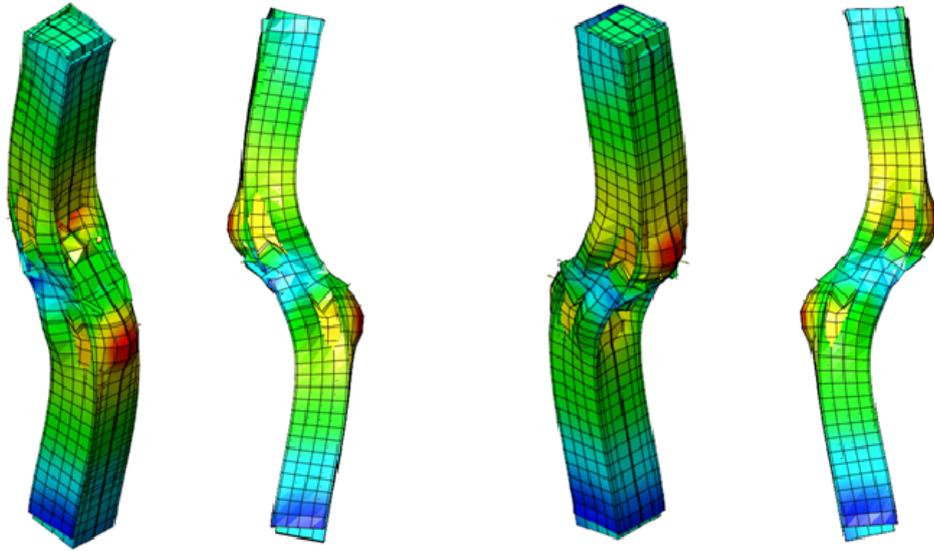


Figure 18: Representation of the deformation of the CFT column.

4 Comparison of modeling and experimental results

To validate the accuracy of the numerical models, the finite element analysis results were compared with the full scale test results of Kataoka [1]. The comparison of the analytical and experimental results showed that the three-dimensional numerical model represented the behavior of the beam-column connection in an appropriate manner.

Comparison of analytical and experimental results showed that the three dimensional numerical model represented the behavior of the connection in a better manner than the two-dimensional model, although the two-dimensional force *versus* displacement curve to be closer the experimental curve. This fact was due to the inclusion of certain details of the connection (full model, interface tube /concrete core) that were incorporated into three-dimensional model that made all difference in stress distribution in the structure.

In the Figure 19 is shown the two-dimensional and three-dimensional force *versus* displacement curve, where the difference between initial stiffness can be observed. The two-dimensional model presented initial stiffness greater than three-dimensional, but with the loading cycles the stiffness became lower.

The comparison between numerical and experimental results showed that to the initial stiffness of the numerical models represented so closely the behavior of the connection, but in the last cycle in the test, the displacements obtained were upper than the numerical results, with residual displacement. The final displacements obtained in the test were higher than expected in numerical simulations due to the appearance of gaps in the reaction structure caused by loading cycles. The curves are shown in Figure 20.

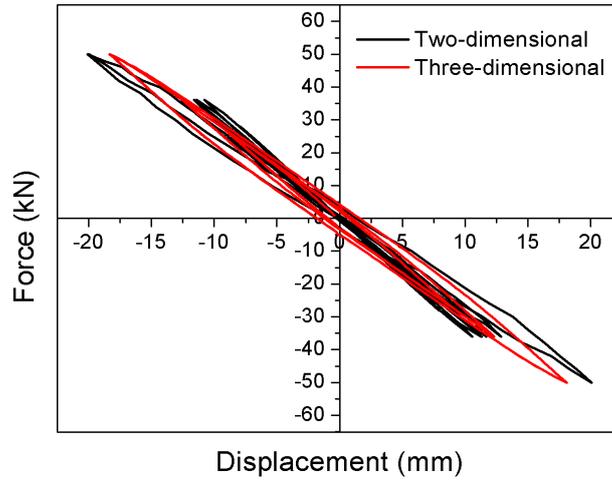


Figure 19: Comparison between two-dimensional and three-dimensional force *versus* displacement curves.

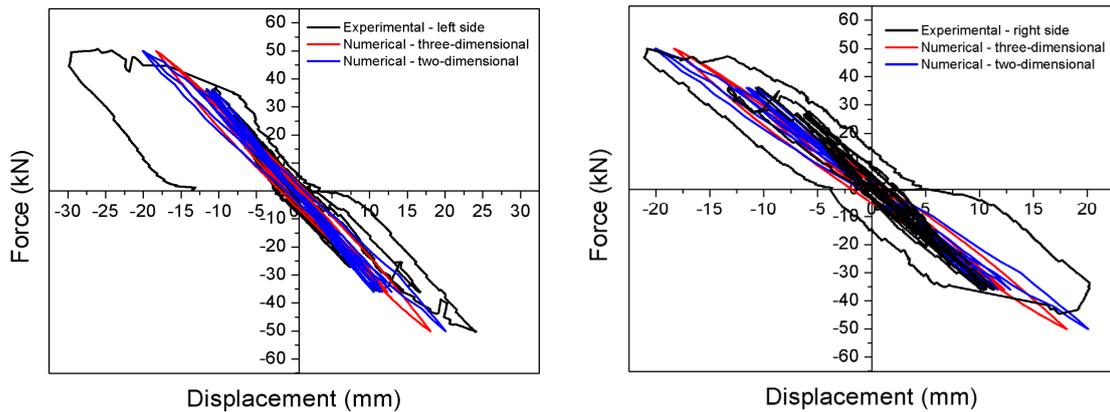


Figure 20: Comparison between numerical and experimental force *versus* displacement curves.

Another comparison was made regarding the stresses developed in the steel elements and in the bolts. In the two-dimensional model both elements reached the yielding stress and in the three-dimensional model just the beams did that. In the test it was not measured the strain in the bolts to determinate the stress, but the results obtained in the three-dimensional model make sense because of the distortion of the column, discussed in many papers about steel and composite connections.

In general, the development of criteria for a certain connection involves first identifying possible failure modes and then establishing appropriate design checks to prevent undesirable failure modes. The results of many experimental studies have shown that failure could take place in either one of following modes: (a) column failure; (b) beam failure, or (c) connection failure (bolts). In this case, the failure

occurred in the beam as indicated in the simulation of the three-dimensional numerical model and in the test. In Figure 21 is shown the beams strain obtained in the test. The values indicate that the beams reached the yielding stress.

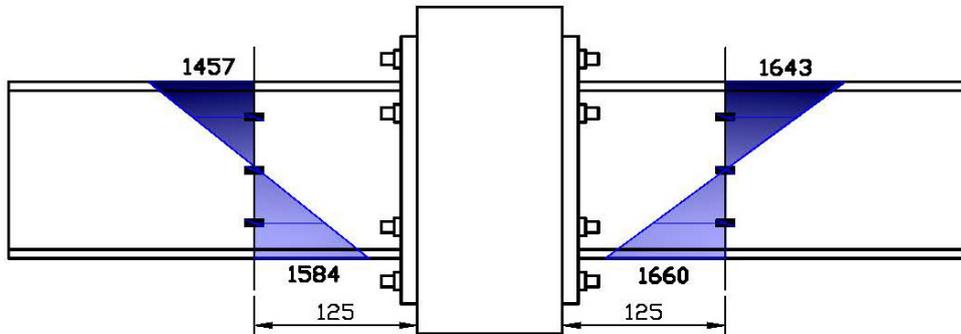


Figure 21: Beams strain in the test (unit: mm/mm x 10⁻⁶) (Kataoka [1]).

5 Conclusion

Based in this study, it can be concluded that the proposed three-dimensional finite element modelling can accurately represent the main features of the behaviour of composite endplate connections. The two-dimensional model does not represent correctly some important parameters to correct understanding of the failure mechanism although to express satisfactorily the force *versus* displacement relationship. From the analysis of the tests and simulations, the following conclusions can be drawn:

- (1) For this type of composite connection the three-dimensional numerical model is recommended to represent the behaviour, because it can show the distortion of the CFT column. In the three-dimensional model the correct geometry of the column was released with the introduction of the interface between the concrete and the steel tube that contributed to approach to the behavior obtained in the test.
- (2) To describe the behaviour of the cruciform connection it is recommended to use a complete model, mainly in studies involving reversible cyclic loading with opposite direction forces applied in each beam.
- (3) The failure occurs in the beams when they reached the yielding stress, according to the three-dimensional simulation and the test. The bolts in these cases were subjected to low forces according to the CFT column distortion. The analytical method was unfeasible to determine the stress in the bolts arising from the behaviour of the column.

- (4) In structures without mix of materials and so many interfaces, like reinforced concrete structure, the two-dimensional model can represent the behavior as good as well, as mentioned in [2, 3].

Acknowledgments

The authors would like to grateful the financial support from FAPESP and the Laboratory of Structures of the University of São Paulo that performed the test of the prototype.

References

- [1] M.N. Kataoka, “Estudo do comportamento de ligações viga-pilar preenchido submetidas a ações cíclicas”, Ph.D. Thesis, University of São Paulo, São Carlos, SP, Brazil, 2011.
- [2] M.N. Kataoka; L.A. Oliveira Júnior; M.A. Ferreira, “Análise Numérica de Ligações Viga-Pilar em Estruturas Pré-Moldadas de Concreto”, 52º Congresso Brasileiro do Concreto - CBC2009. Curitiba, PR, Brazil, 2009.
- [3] M.N. Kataoka; L.A. Oliveira Júnior; M.A. Ferreira, “Análise numérica de uma ligação viga-pilar em estrutura pré-moldada de concreto sob a ação de forças cíclicas”. 52º Congresso Brasileiro do Concreto - CBC2010. Fortaleza, CE, Brazil, 2010.
- [4] J.W. Hu; R.T. Leon, “Analyses and evaluations for composite-moment frames with SMA PR-CFT connections”. Springer Science. v.65, p. 433 – 455.
- [5] H.L. Soriano, “Método de Elementos Finitos em Análise de Estruturas”. Editora da Universidade de São Paulo. São Paulo, SP, Brazil, 2003.
- [6] TNO Building and Construction Research. “Diana User’s Manual – Release 9, Delft”, Netherlands. 622 p, 2005.