

**Influence of Modelling Issues on
Nonlinear Static Seismic Analysis of
a Regular 3D Steel Structure**

A. Belejo; R. Bento
- Maio de 2012 -

**Relatório ICIST
DTC nº 13/2012**

LIST OF CONTENTS

LIST OF CONTENTS.....	2
1 INTRODUCTION	3
2 STRUCTURE DESCRIPTION.....	4
3 MODELLING.....	5
4 ACTION.....	6
5 NUMERICAL STUDY – RESULTS.....	6
5.1 Dynamic Characteristics	6
5.2 Pushover Curves.....	7
5.3 Comparison and Analysis of all Pushover Curves	10
5.4 Seismic Assessment using N2 Method	11
5.5 Nonlinear Dynamic Analyses	16
6 CONCLUSIONS	18
REFERENCES	19
ACKNOWLEDGMENTS.....	20

1 INTRODUCTION

In the scope of the research project PTDC/ECM/100299/2008 (www.3disp.org), and as far as Task 2 is concerned (evaluate the influence of modelling issues on nonlinear seismic static analysis of 3D irregular structures), a steel structure was selected and studied, in order to be assessed in terms of seismic performance. The seismic assessment was performed through nonlinear static procedures using different structural analysis software and the results obtained were compared. The structural programs herein used are *SAP2000* v14.2.4 (Computers and Structures, 2010), *SeismoStruct* v5.2.2 (SeismoSoft, 2011) and *OpenSEES* (PEER, 2006).

The two last mentioned software are fibre-based structural analysis programs and are oriented to this type of nonlinear analysis. So it is interesting evaluate the performance of the commercial program *SAP2000*, essentially oriented for structural design, comparing the results obtained with it with the ones achieved by means of *SeismoStruct* or *OpenSEES* software. Therefore pushover curves were obtained, in both directions of a steel structure, by the three structural analysis programs, and the performance of the steel structure is assessed using N2 method.

The objective of the study presented herein is to test some of these models in a regular steel structure and evaluate their precision in nonlinear static and dynamic analyses, with the use of three advanced computer programs: *SAP2000*, *OpenSEES* and *SeismoStruct*.

Other objective of this work is to perform Nonlinear Static Analyses in aforementioned software and evaluate the influence of modelling issues and their precision, specifically in *SAP2000*, in nonlinear analyses. Moreover, based on this study and on the results obtained a training course on Nonlinear Static Analysis (Pushover) of steel structures would be defined and proposed in a near future. This will be based on a former course organized in the scope of this research project (http://www.civil.ist.utl.pt/~rbento/tmp/3DISP/AENLs_FEV2012.pdf), in February 2012, focusing in RC structures and called (“Análises Estáticas não Lineares (Análises *Pushover*) para o Dimensionamento e Avaliação Sísmica de Estruturas”).

The N2 method herein used was proposed by Fajfar (Fajfar and Fischinger, 1998; Fajfar, 2000) and included in EC8 (CEN, 2004) for Nonlinear Static Analysis. Its procedure is described in EC8 - part1, Annex B. Since the structure is plan-regular, the extended N2 Method also proposed by Fajfar (Fajfar et al, 2005) was not used.

The comparison with the results obtained through Nonlinear Dynamic Analysis with *OpenSEES*, which is used as a reference for this type of analysis, allowed to evaluate the accuracy of the results obtained by means of I nonlinear static analyses.

2 STRUCTURE DESCRIPTION

The structure analysed is a regular Steel frame structure, with a 4.50 m height storey.

Figures 1 and 2 represent a plan view and a 3D view of the structure studied.

This structure presents four transversal frames and two longitudinal. Columns HEB180 and beams IPE270 or IPE160 compose the two transversal end frames. The same beams and columns HEB240 compose middle frames. Longitudinal frames, beyond the columns that belong to transversal frames have a HEB240 column between end and middle transversal frames. Their first and last beams are IPE300 and all the others are IPE270. There are some other columns and beams, however they are designed only to support gravity loads.

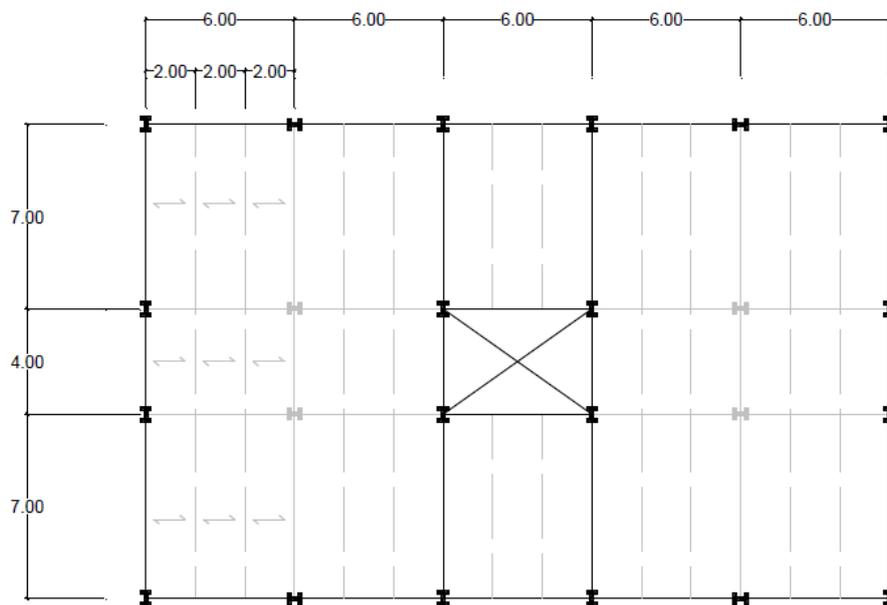


Fig. 1 – Structure Plan View (Peres et al, 2011)

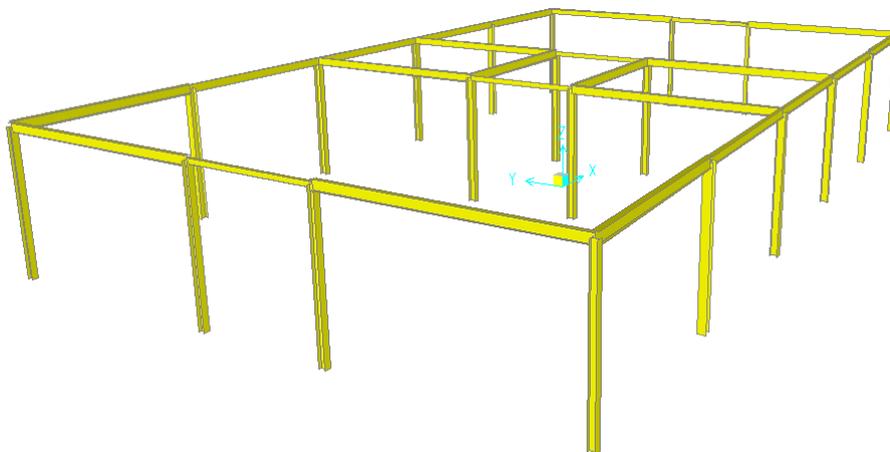


Fig. 2 – Structure 3D-view

3 MODELLING

The structure herein studied was modelled by an assemblage of inter-connected frame elements using centreline dimensions. The material used was S275 steel with a Young's modulus corresponding to 210 GPa. An average yield strength of 275 MPa and an ultimate strength of 430 MPa were assumed for the material. A simplified bilinear stress-strain relationship was used and a value of 1% was considered for the strain hardening (Peres et al, 2010).

There are two different approaches available in *SeismoStruct* and *OpenSEES* to model the elements: the *forced based* and *displacement based* elements formulations. In general, the displacement-based elements assume cubic displacement shape functions and present distributed plasticity. In the other hand the force-based elements assume exact force fields in members without rigid body modes and they have also distributed plasticity with linear curvature distribution. In the work, the choice in both programs was forced based elements instead displacement based elements and it was related with the fact this formulation is "exact" within the beam theory (Peres et al, 2010).

In *OpenSEES* and *SeismoStruct*, material inelasticity is distributed along the entire structural elements.

In SAP2000, although the concentrated plasticity (lumped models) is the only choice in nonlinear modelling, different ways of modelling the nonlinearity of the structure were tested to obtain the capacity curve of the structure. The nonlinearity was considered adopting Plastic Hinges with hysteretic relationships based on FEMA-356 (FEMA, 2000) tables or applying Plastic Hinges with fibre elements, where a material constitutive relationship was assigned to any fibre Plastic Hinges with hysteretic relationships based on FEMA356 tables are automatically applied. Thus, the latter option is much easier to be applied by the user.

The rigid in-plane stiffness of the floor slab was modelled considering a rigid diaphragm where nodal displacements can be expressed by three rigid body motions, namely two horizontal translations and one rotation about the normal to the floor-plane. In *SeismoStruct*, rigid diaphragm was modelled with *Lagrange Multipliers* as Constraint type.

A lumped mass modelling strategy was adopted, in which masses were lumped at the nodal points according to its tributary area or where secondary elements are uncharged. Total translational masses amounted to 206.5 tones.

In *OpenSEES* and *SeismoStruct*, the software computed automatically the vertical gravity loads, using the masses previously defined. In *SAP2000* the opposite process was applied, firstly the gravity loads were defined and the masses were automatically computed afterwards.

Modelling of fibre sections in SAP2000

There are some different methods to use Plastic Hinges by means of fibre discretization in SAP2000. Fibres can be generated automatically choosing “Default from section” option or applying “Section Designer” tool and draw all the sections, while fibres are also automatically defined. In both approaches, 7 and 9 fibres in each section were defined respectively. The process to define more fibres is defining them manually.

4 ACTION

The seismic action was defined assuming an Earthquake Type 1 and a soil type B (EC8). Therefore, the values recommended for the soil factor and the spectral periods are: $S=1.2$, $TB=0.15$, $TC=0.50$ and $TD=2.0$. The seismic intensity considered was $ag = 0.3g$.

For this study, seven bi-directional real ground motion records, presented in Table 1, were selected by Peres, R (Peres, R et al, 2012) and fitted to EC8 elastic response spectrum previously mentioned. These seven bi-directional accelerograms were used for the Nonlinear Dynamic Analyses.

Table 1 – Ground Motion records considered

Earthquake Name	Station ID	Waveform ID
Montenegro 1979	ST63	000197
Umbria Marche 1997	ST228	000612
Montenegro (aftershock) 1997	ST77	000232
Racha (aftershock) 1991	ST200	000530
Dinar 1995	ST543	001720
Campano Lucano 1980	ST99	000293
Montenegro 1979	ST62	000196

5 NUMERICAL STUDY – RESULTS

In this study, before performing pushover analyses, dynamic characteristics of the structure were defined and analysed.

5.1 Dynamic Characteristics

To confirm that models represent the same structure in all programs, modal analyses were performed and periods of vibration obtained by all the programs. The values obtained are presented in Table 2, for comparison.

Table 2 – Three first mode Periods on each program

Mode	Program	Period (sec)
1	SAP2000	0.72
	SeismoStruct	0.74
	OpenSEES	0.72
2	SAP2000	0.66
	SeismoStruct	0.68
	OpenSEES	0.66
3	SAP2000	0.47
	SeismoStruct	0.48
	OpenSEES	0.47

As it shown, for first three modes the periods of vibration are similar. The same was obtained for the lower periods (higher frequencies) and for the correspondent modes of vibration.

5.2 Pushover Curves

In all approaches, pushover curves were obtained until a roof drift of 3% was reached. The monitored node was assigned to centre of mass of the top floor.

All pushover curves obtained (for different models and different software) and the nonlinear dynamic results (maximum displacements and corresponding Base Shear) for an intensity level of 0.3g obtained with *OpenSEES* are presented in this section.

Firstly, nonlinear behaviour of the hinge zone was defined by means of a force-deformation model suggested in **tables 5.6 of FEMA356** was used in **SAP2000**. Fig. 3 shows the pushover curves in both directions for this approach and the nonlinear dynamic results aforementioned.

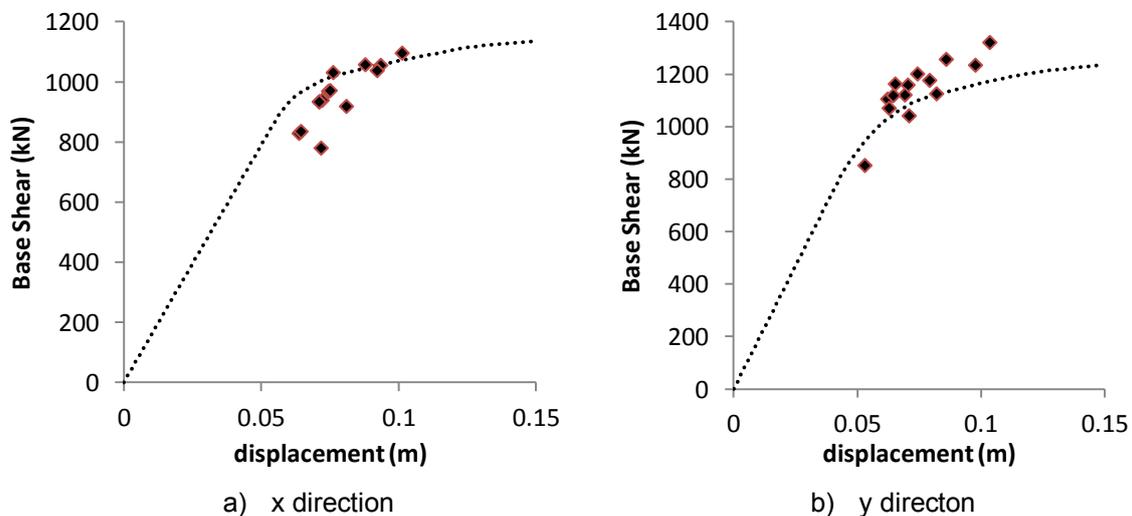


Fig. 3 - Pushover Curves from SAP2000 using a force-deformation model suggested in FEMA356

Figure 3 shows a reasonable match between pushover curves and nonlinear dynamic analyses results.

Pushover curves obtained in **SAP2000** considering the inelastic behaviour through Plastic Hinges with **fibre elements** with the approaches already mentioned (“Default from section” option”, applying Section Designer or defining all fibre (manually) were obtained and are shown in Fig. 4 for x directions and in Fig. 5 for y direction.

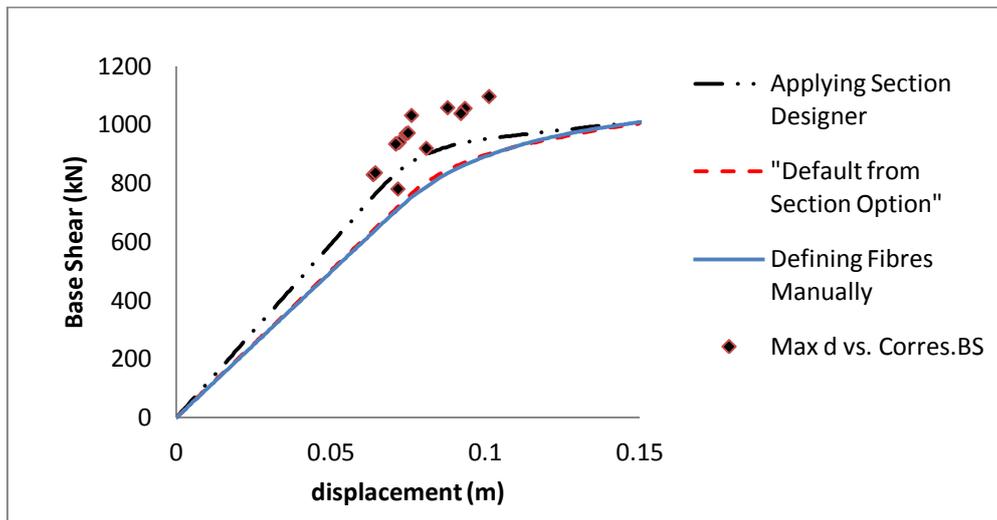


Fig. 4 - Pushover Curves for x direction associated to SAP2000 according with “Default from section” option

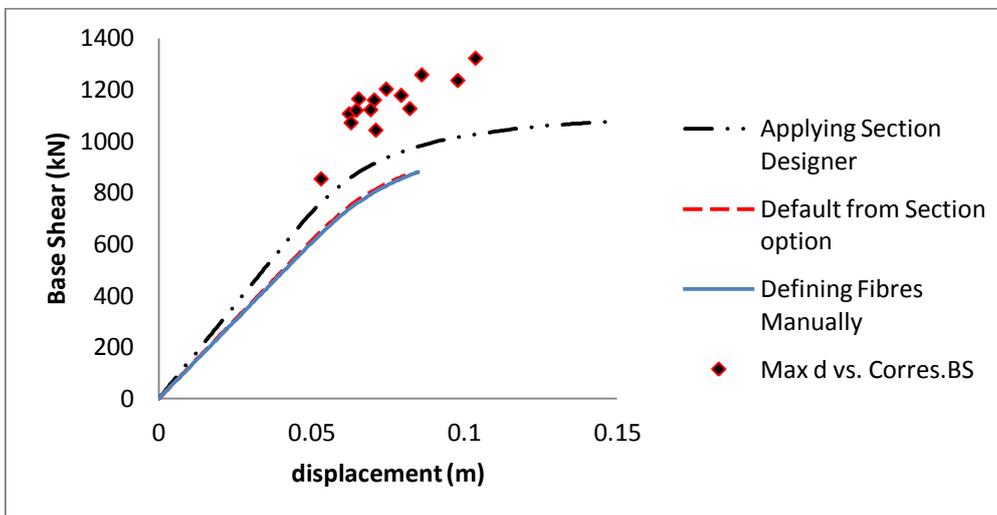


Fig. 5 - Pushover Curves for y direction associated to SAP2000 according with “Default from section” option

Only applying Plastic Hinges with fibre elements using Section Designer, complete results were obtained for both directions. Otherwise pushover curves for y direction were not able to reach the value defined for the monitored displacement. And in both directions can be seen that the results with nonlinear dynamic analyses do not match the pushover curves, being above them.

Analysing only the pushover curves obtained in *SAP2000* and comparing their position with respect to the results from Nonlinear dynamic analyses one can be concluded that for this structure, hinge zone was defined by means of a force-deformation model suggested in tables 5.6 of FEMA356 reach to closer results.

Performing nonlinear static analysis with *SeismoStruct* with all considerations previously mentioned, the pushover curves for both directions were obtained and depicted in the following.

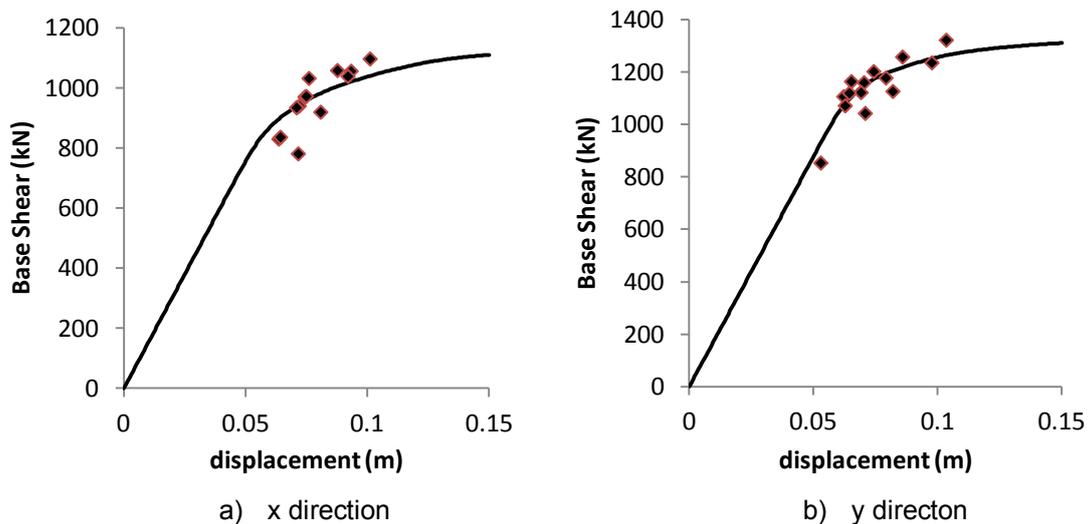


Fig. 6 - Pushover Curves for x and y directions associated to *SeismoStruct*

Herein, can be observed that pushover curves obtained with *SeismoStruct* perfectly match with nonlinear dynamic analyses results.

The structure was also analysed with *OpenSEES* and pushover curves for x and y directions are plotted in Fig. 7.

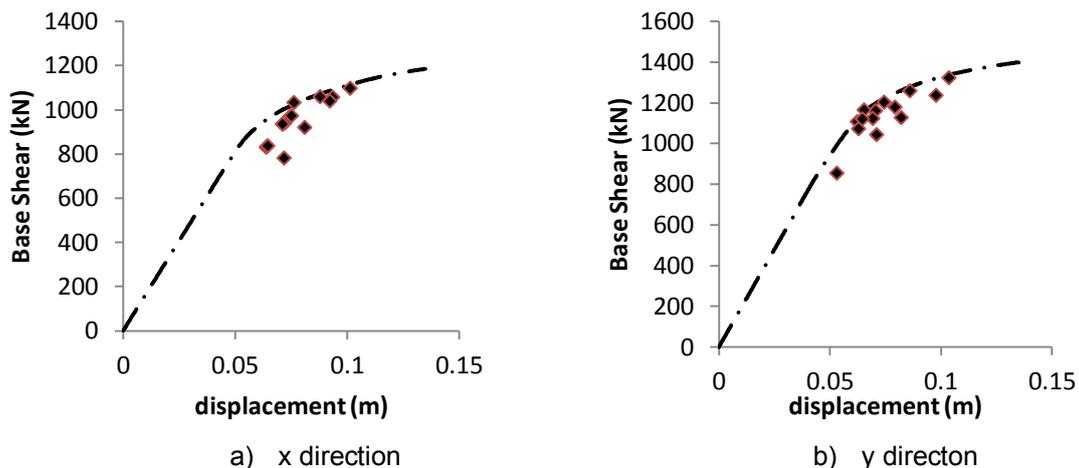


Fig. 7 – Pushover Curves for x and y directions associated to *OpenSEES*

And it is shown also a reasonable match between pushover curves and a Nonlinear Dynamic Analyses results.

5.3 Comparison and Analysis of all Pushover Curves

Pushover curves obtained through all programs are defined in the same graph for each direction (Fig. 8 and 9). As far as SAP2000 is concerned only Pushover Curves obtained by means of Section Designer were plotted.

Thus pushover curves obtained in *SAP2000*, *SeismoStruct* and *OpenSEES* can be compared.

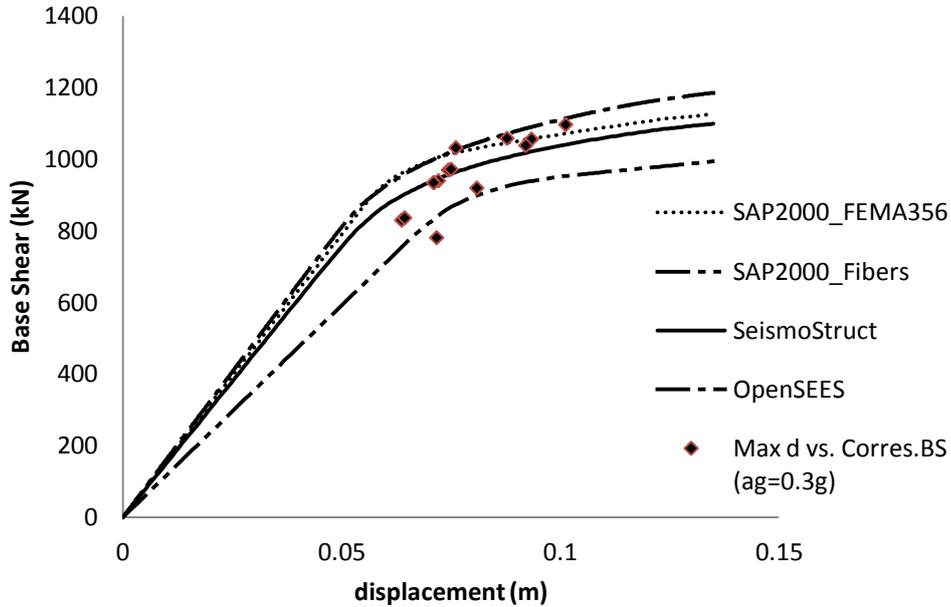


Fig. 8 - Pushover Curves obtained for x direction and associated to each software

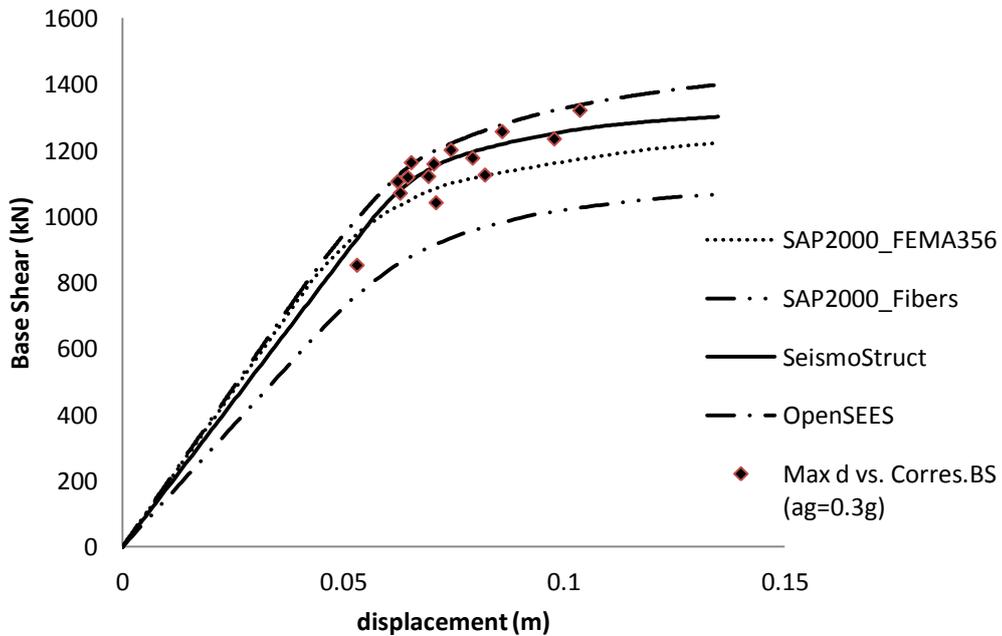


Fig. 9 - Pushover Curves obtained for y direction and associated to each software

From the results obtained in terms of Pushover curves, it seems that *SAP2000* is able to be a useful tool for seismic assessment of steel buildings using Nonlinear Static procedures.

The curves confirm that a force-deformation model suggested in tables 5.6 from FEMA356 to define Plastic Hinges leads to better results than Fibre elements, showing also reasonably satisfactory results comparing with those obtained by *SeismoStruct* or *OpenSEES*. This idea is enhanced by the fact that Pushover Curve obtained with *SAP2000* through force-deformation models suggested in FEMA356 is exactly between the curves associated to the reference programs for x direction and close for y direction.

5.4 Seismic Assessment using N2 Method

In order to achieve the structure performance point (target displacement), N2 Method was applied for pushover curves obtained with all programs. For all cases, was verified that target displacement obtained was much different from the displacement correspondent to the plastic mechanism (and from which the first idealized elasto-plastic force-displacement relationship is defined). So an iterative process was applied as is suggested in as EC8-part1. The graphical representation for all cases is showed in Tables 3, 4, 5 and 6. Finally, Table 7 resumes the target displacements obtained and corresponding Base Shear values.

The meaning of the current symbols in the execution of N2 method is listed in the following:

F^* - SDOF system base shear

d^* - SDOF system displacement

F_y^* - Yield strength of the idealized bilinear SDOF system

d_y^* - Yield displacement of the idealized bilinear SDOF system

S_{ae} - Elastic acceleration response spectrum

d_t - Target displacement of the MDOF system

Table 3 – N2 Method applied with pushover curve obtained in *SAP2000* (Computers and Structures, 2010) with plastic hinges through Tables from FEMA356

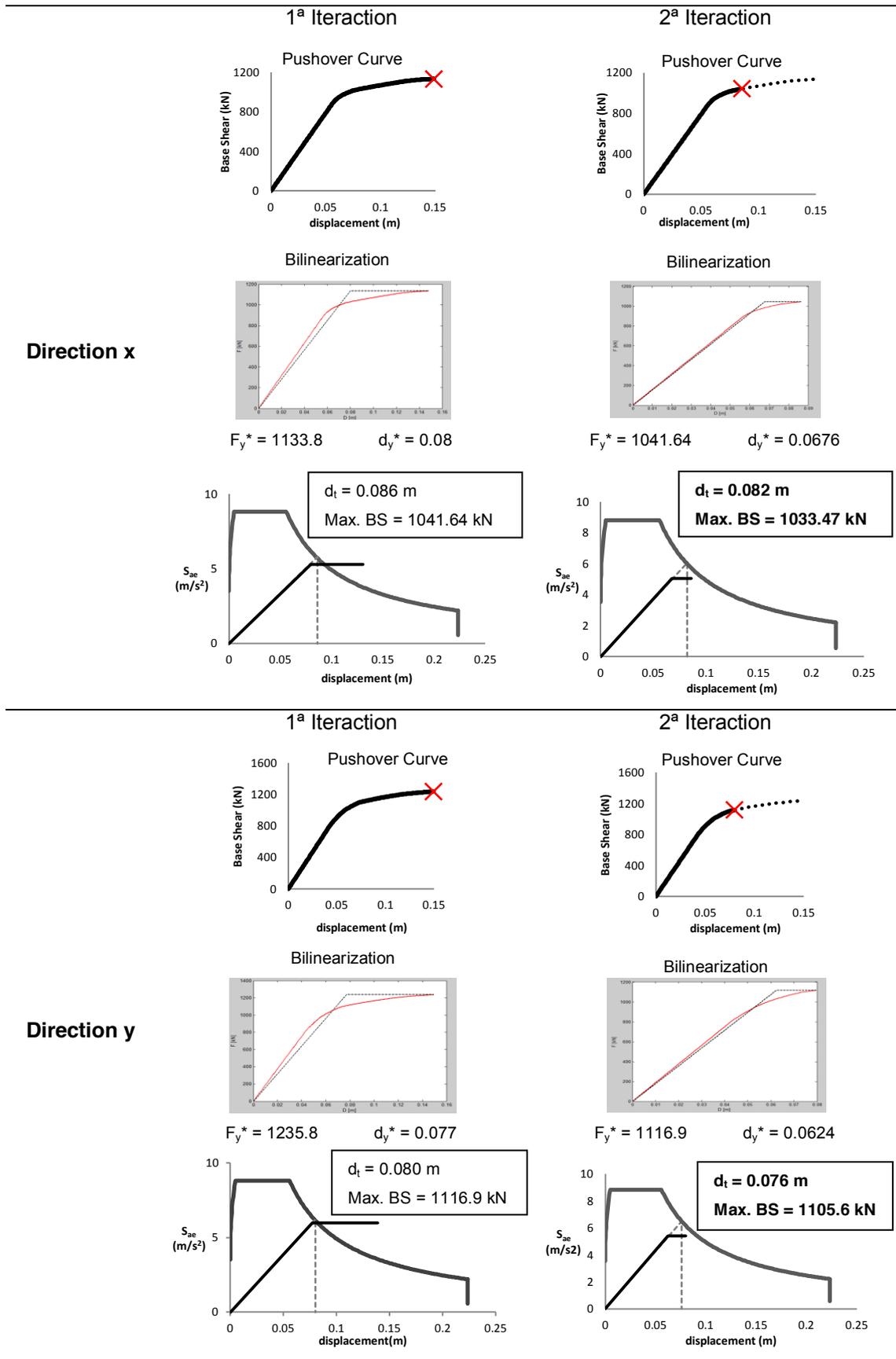


Table 4 – N2 Method applied with pushover curve obtained in *SAP2000* (Computers and Structures, 2010) with plastic hinges through fibre elements

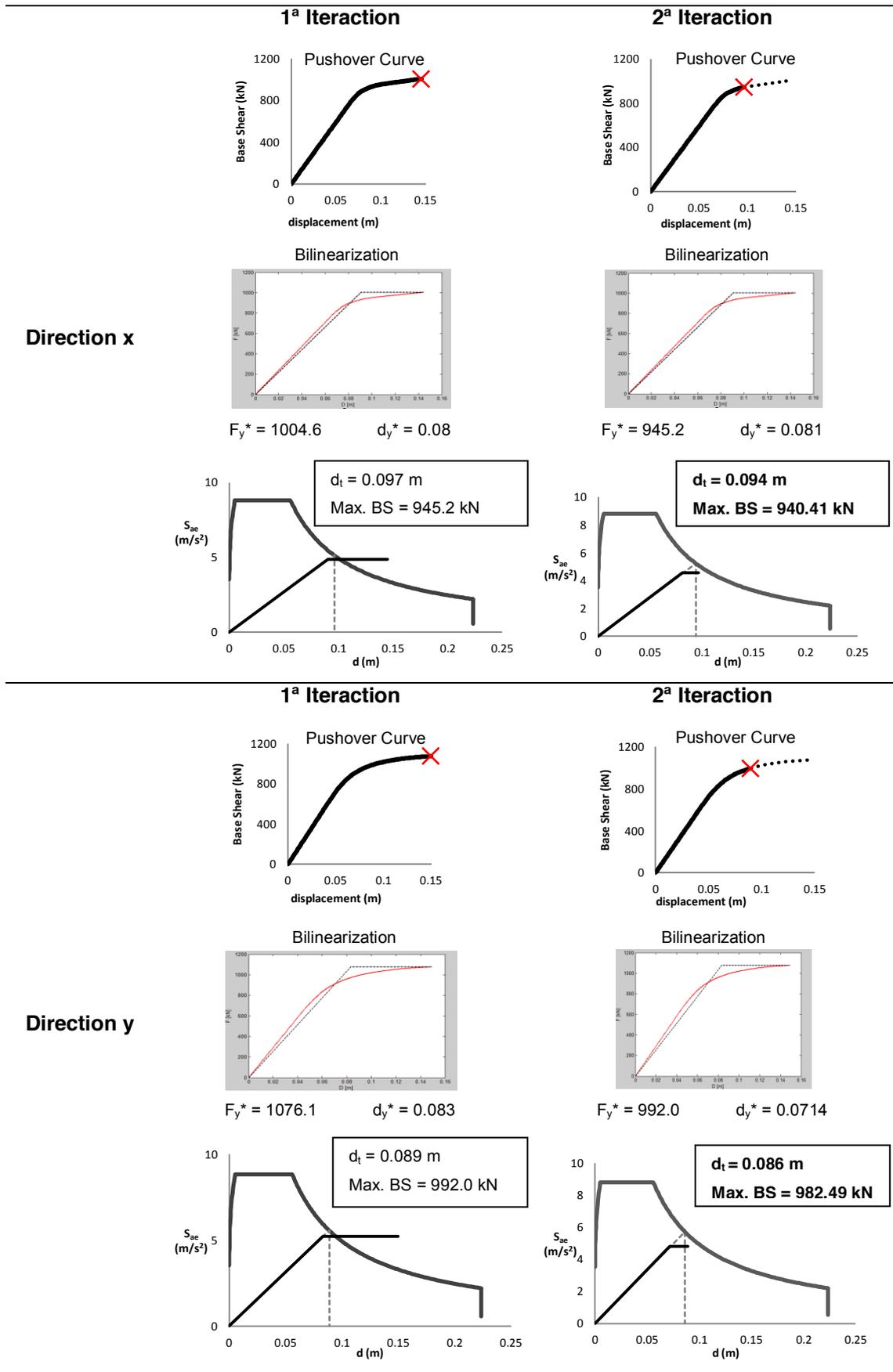


Table 5 - N2 Method applied with pushover curve obtained in *SeismoStruct* (Seismosoft, 2011)

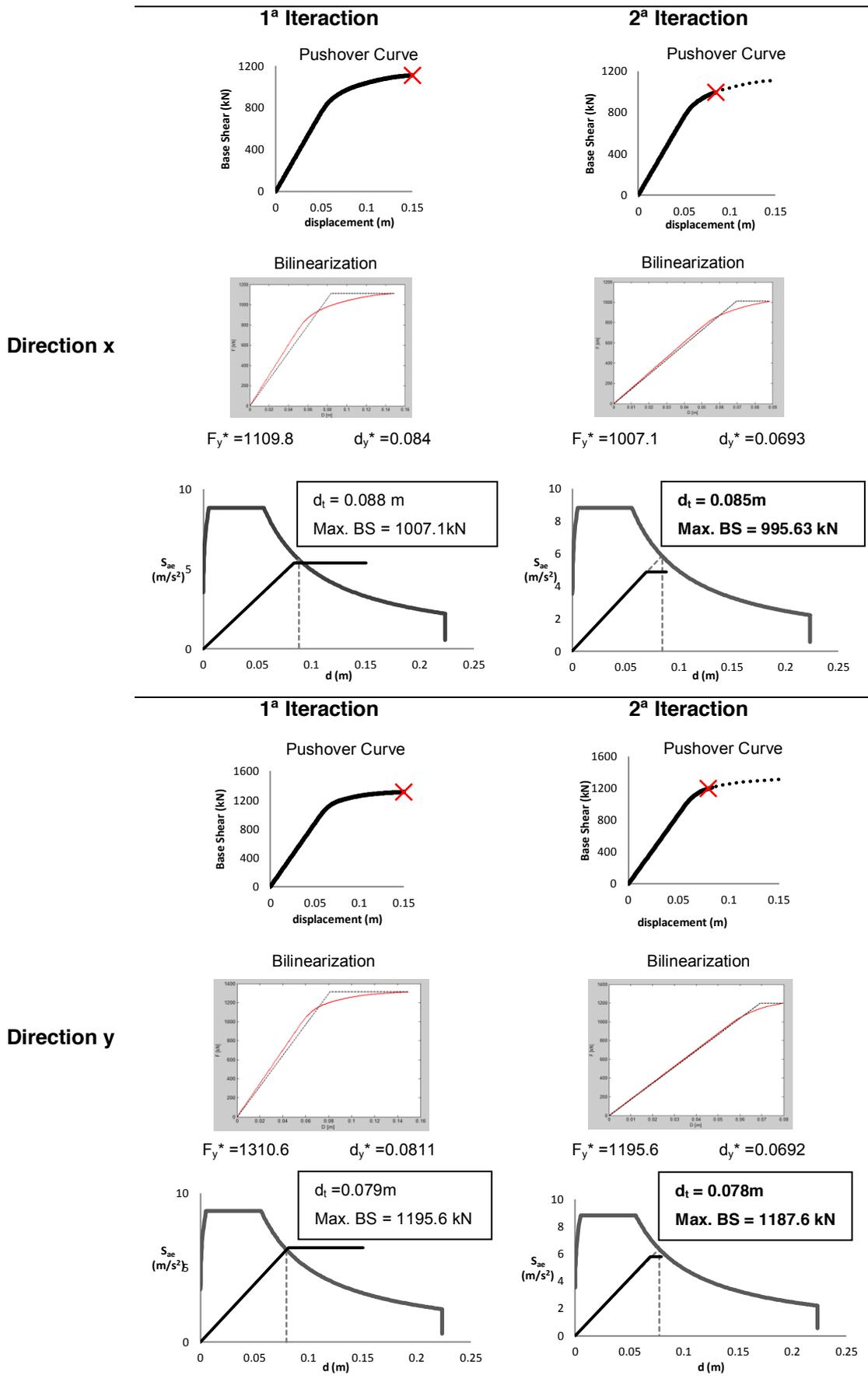


Table 6 – N2 Method applied with pushover curve obtained in *OpenSEES* (PEER, 2000)

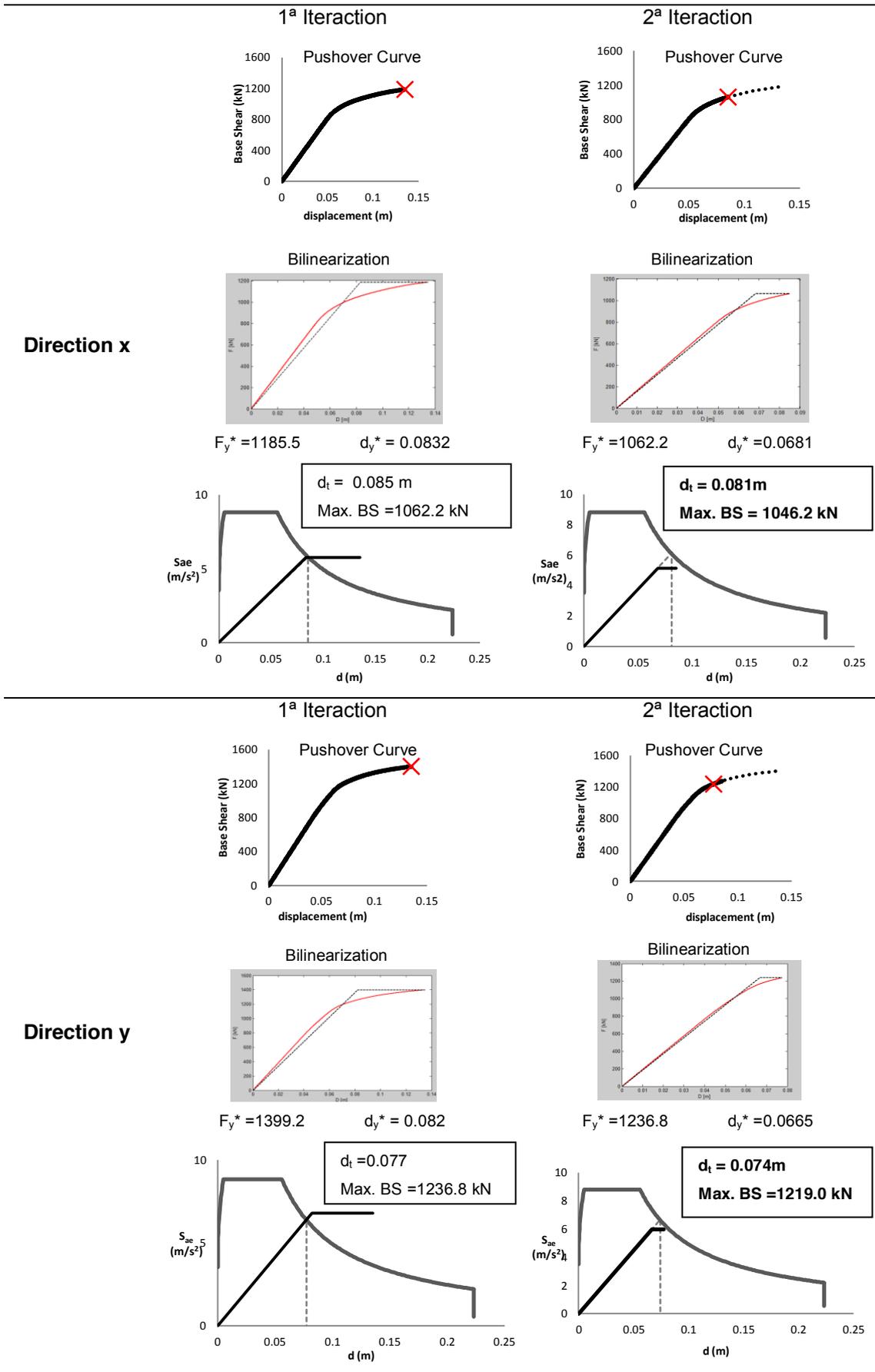


Table 7 – Performance point obtained with each approach or program

Software/Approach	Direction x		Direction y	
	Target Displacement (m)	Max. Base Shear (kN)	Target Displacement (m)	Max. Base Shear (kN)
<i>SAP2000</i> – tables from FEMA356	0.082	1033.5	0.076	1105.6
<i>SAP2000</i> – fibre elements	0.094	940.4	0.086	982.5
<i>SeismoStruct</i>	0.085	995.6	0.078	1187.6
<i>OpenSEES</i>	0.081	1046.2	0.074	1219.0

As previously referred, for the N2 method it was performed a second iteration for all cases following the suggestion proposed by EC8, because the target displacement d_t is much different from the ultimate displacement defined to obtain the bilinear SDOF curve.

The results herein obtained with the proposed static procedure confirm what was previously mentioned. The model suggested in FEMA356 to define Plastic Hinges leads to satisfactory results comparing with those obtained by *SeismoStruct* or *OpenSEES*.

The approach where Plastic Hinges are defined as Fibre elements does not lead to so good results as the model suggested in FEMA356.

However the best way to confirm all these assumptions is to compare the results obtained with Nonlinear Dynamic analyses.

5.5 Nonlinear Dynamic Analyses

As previously mentioned, Nonlinear Dynamic Analyses were performed, with a suit of seven bi-directional accelerograms, properly selected as previously referred, using *OpenSEES*.

All response spectra due the aforementioned set of seven semi-artificial ground motions, in both directions was obtained and fitted to the same response spectrum as is show in Fig. 11.

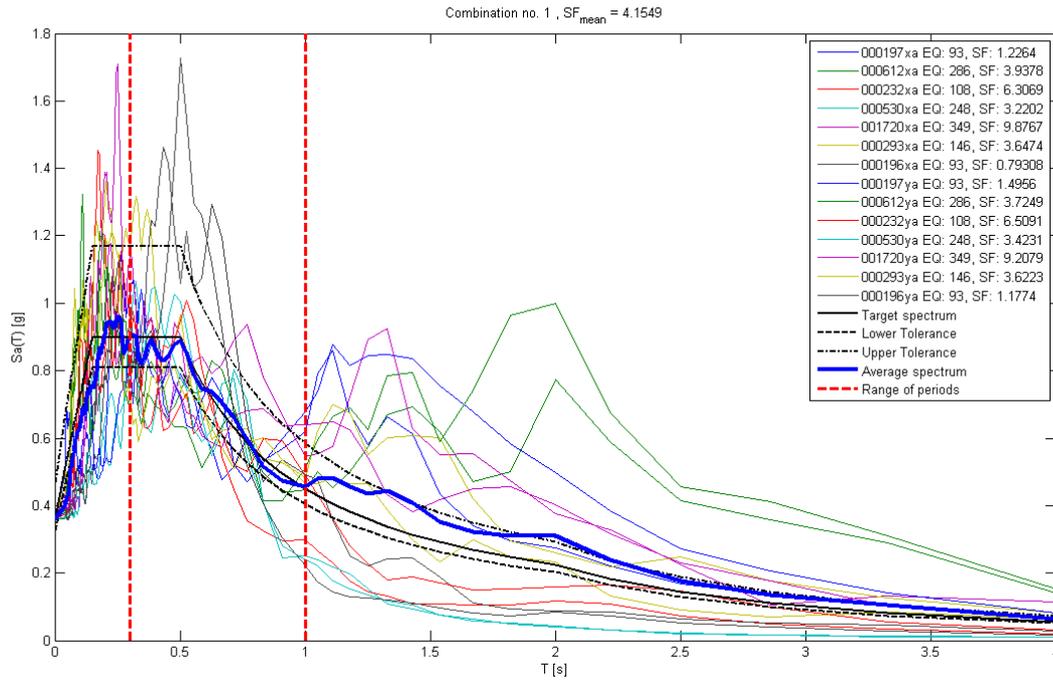


Fig. 10 - Average of all Response Spectrums according fitted to EC8 Spectrum (Peres, R et al, 2012)

The nonlinear dynamic analyses were performed simultaneously in both directions of the structure. Therefore, two analyses were carried out: XxYy (X component according to x direction of the structure and Y component according to y direction of the structure) and XyYx (X component according to y direction of the structure and Y component according to x direction of the structure), for an intensity level of 0.3g.

The mean response among 14 analyses was considered as the 'true' response of the structure.

For each record, the higher displacements and the corresponding Base Shear from Nonlinear Dynamic Analyses are presented in the following Table 8 for both directions, as well as the average results.

Table 8 – Displacements a corresponding Base Shear obtained in Nonlinear Dynamic Analyses

Record	XxYy				XyYx			
	dx max (m)	BSx max (kN)	dy max (m)	BSy max (kN)	dx max (m)	BSx max (kN)	dy max (m)	BSy max (kN)
000196	0.0934	1055.7	0.0705	1159.4	0.0717	780.41	0.1036	1321.7
000197	0.0809	919.17	0.0821	1126.0	0.0711	934.47	0.0629	1071.0
000232	0.0637	829.46	0.0793	1177.3	0.075	971.78	0.0654	1163.6
000293	0.0878	1057.9	0.0693	1121.6	0.0644	835.69	0.0646	1119.2
000530	0.0744	969.5	0.0623	1106.1	0.071	934.11	0.0531	852.9
000612	0.0721	939.56	0.071	1042.2	0.0921	1038.2	0.086	1257.4
001720	0.1012	1096.3	0.0744	1201.8	0.0761	1031.6	0.0978	1235.2
	dx (mean)			0.078 m	BSx (mean)			956.7 kN
	dy (mean)			0.074 m	BSy (mean)			1139.7 kN

In Table 9 were included the results obtained through a Nonlinear Dynamic Analyses as well as in Table 7, and taking them as reference values, some assumptions can be more presented.

Table 9 – Target displacement and maximum Base Shear obtained with each approach or program and Nonlinear Dynamic Analyses

Software used/Approach	Direction x		Direction y	
	Target Displacement (m)	Max Base Shear (kN)	Target Displacement (m)	Max Base Shear (kN)
<i>SAP2000</i> – tables from FEMA356	0.082 (5.1%)	1033.5	0.076 (2.7%)	1105.6
<i>SAP2000</i> – fibre elements	0.094 (20.5%)	940.4	0.086 (16.2%)	982.5
<i>SeismoStruct</i>	0.085 (9.0%)	995.6	0.078 (5.4%)	1187.6
<i>OpenSEES</i>	0.081 (3.8%)	1046.2	0.074 (0.0%)	1219.0
Nonlinear Dynamic Analyses	0.078	956.7	0.074	1139.7

The comparisons with the results obtained with Nonlinear Dynamic Analyses, for this case study, seemed to show that Pushover Analyses proceeded with *OpenSEES* or *SeismoStruct* lead to satisfactory results. Therefore the previously assumption that also satisfactory results are obtained through Pushover Analysis in *SAP2000*, considering the model suggested in FEMA356 to define Plastic Hinges seems to be adequate for this type of structures.

6 CONCLUSIONS

The Nonlinear Static procedure used in seismic assessment of the Regular Steel Structure was the N2 method, and its effectiveness was confirmed for the case study, as it was expected due the regularity of the structure.

The results herein obtained, applying the N2 method and using the pushover curves previously defined through all programs are reasonably close to the results of Nonlinear Dynamic Analyses.

Although *SAP2000* being a commercial software package used essentially for the design of structures, it can be also a useful tool for nonlinear static analysis. Therefore a Training Course should be very useful to teach design engineers to perform nonlinear static analyses with *SAP2000* for the seismic design and assessment of steel structures.

Based on previous and the current studies perform in scope of 3DISP project (www.3disp.org), one can suggest to use the abovementioned version of *SAP2000* to perform Nonlinear Static Analysis. Moreover, it is recommended to choose the approach where plastic hinges are defined by using a force-deformation model suggested in tables 5.6 from FEMA356, instead of fibre sections. In addition, to be easier and more convenient this approach, pushover curves and results obtained with N2 method are similar to the results obtained with the other programs and also Nonlinear Dynamic Analyses.

Nevertheless, given the preliminary nature of this study for steel structures, additional work considering irregular 3D buildings, multi-storey buildings and even multi-storey irregular buildings must be carried out before any definitive conclusion and recommendation about the accuracy of *SAP2000* for the seismic assessment of steel structures by means of Pushover analyses.

REFERENCES

Computers and Structures. 2010; Inc., CSI Analysis Reference Manual for SAP2000, ETABS and SAFE, Computer and Structures, Inc., Berkley, California

CEN, Eurocode 8 (EC8) 2004: Design of structures for earthquake resistance. Part 1: general rules, seismic actions and rules for buildings. EN 1998-1:2004 Comité Européen de Normalisation, Brussels, Belgium.

Fajfar P., Fischinger M. 1998: N2 - A method for non-linear seismic analysis of regular buildings, Proceedings of the Ninth World Conference in Earthquake Engineering. pp. 111-116, Tokyo-Kyoto, Japan

Fajfar P., Marusic D., Peris I. 2005a: The extension of the N2 method to asymmetric buildings. In: Proceedings of the 4th European workshop on the seismic behaviour of irregular and complex structures, CD ROM. Thessaloniki, August 2002

FEMA356 2000: Prestandard and Commentary for the Seismic Rehabilitation of Buildings, ASCE, American Society of Civil Engineers, Washington D.C.

PEER, 2006 OpenSEES: Open System for Earthquake Engineering Simulation. Pacific Earthquake Engineering Research Center, University of California, Berkeley, CA.

Peris R., Bento R., Castro M, 2010: Modelling Issues, PTDC/ECM/100299/2008 – Report, available in <http://www.3disp.org>

Peres R., Bento R., Castro M, 2011: Design Procedures, PTDC/ECM/100299/2008 – Report, available in <http://www.3disp.org>

Peres, R., Bento R., 2012: Performance Base Design and Assessment of Irregular Steel Structures.

Seismosoft, 2011, “SeismoStruct – A computer program for static and dynamic nonlinear analysis of framed structures“ available online from <http://www.seismosoft.com>.

ACKNOWLEDGMENTS

The authors would like to acknowledge the financial support of the Portuguese Foundation for Science and Technology through the research project PTDC/ECM/100299/2008. In addition, the authors would like to thank Rita Peres, researcher in the same project, who performed all work about Nonlinear Dynamic Analysis presented in this report.