View metadata, citation and similar papers at core.ac.uk

brought to you by CORE

provided by Archivio istituzionale della ricerca - Politecnico di Milano

COMPDYN 2015
5th ECCOMAS Thematic Conference on
Computational Methods in Structural Dynamics and Earthquake Engineering
M. Papadrakakis, V. Papadopoulos, V. Plevris (eds.)
Crete Island, Greece, 25–27 May 2015

# INELASTIC CYCLIC NUMERICAL ANALYSIS OF STEEL STRUTS USING DISTRIBUTED PLASTICITY APPROACH

**Alper Kanyilmaz**[1]

[1]Department of Architecture, Built Environment and Construction Engineering
Politecnico di Milano
Via Ponzio 31, 20133 Milano, Italy
alper.kanyilmaz@polimi.it

**Keywords:** Distributed plasticity, inelastic cyclic modeling, fiber based beam elements

**Abstract:** *This paper presents a comparison between two different modeling approaches: Refined FE modeling using volumetric elements, and fiber modeling using beam elements with distributed plasticity. The numerical models calibrated with the experimental result from existing literature, reproduce the behavior of cold formed SHS40, and hot rolled DUPE100 steel elements under inelastic cyclic loading. The hysteresis loops obtained from two models show that the accuracy obtained by simpler fiber-element formulation is quite close to the more refined volumetric model. In terms of computation time, distributed plasticity model is much more efficient, and can be a good option to perform nonlinear analysis of multi-level buildings, which would be quite cumbersome with volumetric modeling approach. This study has been realized thanks to the research fund received from European commission with the contract MEAKADO RFSR-CT-2013-00022.*

# 1. MODELING OF CYCLIC BEHAVIOUR OF BRACING ELEMENTS

Inelastic deformation of bracing elements is the main parameter affecting the seismic performance of braced frames during a seismic event.

The objective of this study is to explore accurate and time efficient modelling ways to simulate inelastic behaviour of steel bracings under cyclic loading by means of validation studies based on experimental data and refined finite element models.

Validation studies shown in this report has been performed in two stages:

i) Numerical model of various steel struts made of cold formed square hollow sections have been developed and calibrated with the inelastic cyclic load tests carried out by [1]. Continuum finite element modelling approach is used (CFEM)

ii) Fiber based distributed plasticity modelling approach has been validated against calibrated continuum finite element model of steel struts (DPE).

In order to obtain experimental data to validate numerical models, Goggins et. al have performed several cyclic tests on cold-formed square and rectangular hollow steel section bracing elements, according to the provisions of the ECCS (1986) [2]. 1 Fifteen specimens have been tested which are fabricated from 20x20x2.5SHS, 40x40x2.5SHS and 50x25x2.5 RHS sections with two different lengths: intermediate (1100mm) and long (3300mm).

We focus on intermediate (Model 1) and long length (Model 2) steel struts of 40x40x2.5 SHS cross- sections as shown in Table 1. A general layout of the test specimens is shown in Figure 1.

| Test ID | Section Size | $\bar{\lambda}$ | d/t | $F_y$ kN | $\delta_y$ mm | $F_{max}$ kN | $\delta_u$ mm | $\mu_\Delta$ |
|---|---|---|---|---|---|---|---|---|
| Series1 | | | | | | | | |
| CyIS1 | 40x40x2.5 | 0.4 | 13.1 | 100.4 | 2.4 | 112.9 | 20.7 | 8.7 |
| CyIS2 | 40x40x2.5 | 0.4 | 13.1 | 101.5 | 2.4 | 112.4 | 17.8 | 7.5 |
| CyIS3 | 20x20x2.0 | 0.9 | 6.7 | 26.0 | 1.2 | 45.4 | 35.3 | 29.7 |
| CyIS4 | 20x20x2.0 | 0.9 | 6.7 | 29.8 | 1.1 | 46.0 | 22.6 | 20.9 |
| CyIS5 | 50x25x2.5 | 0.6 | 17.1 | 78.1 | 1.6 | 111.5 | 23.0 | 14.3 |
| CyIS6 | 50x25x2.5 | 0.6 | 17.1 | 83.1 | 1.8 | 111.5 | 21.8 | 12.2 |
| Series2 | | | | | | | | |
| CyLS1 | 40x40x2.5 | 1.3 | 12.9 | 123.1 | 4.3 | 143.9 | – | 9.5* |
| CyLS2 | 40x40x2.5 | 1.3 | 12.9 | 124.1 | 4.4 | 143.8 | – | 9.2* |
| CyLS3 | 40x40x2.5 | 1.3 | 12.9 | 126.5 | 5.7 | 142.5 | – | 7.2* |
| CyLS4 | 20x20x2.0 | 3.2 | 6.5 | 63.1 | 7.0 | 70.2 | 16.0 | 2.3 |
| CyLS5 | 20x20x2.0 | 3.0 | 6.0 | 64.4 | 7.9 | 69.3 | 15.2 | 2.0 |
| CyLS6 | 20x20x2.0 | 3.0 | 6.0 | 63.6 | 6.0 | 72.3 | 16.7 | 2.8 |
| CyLS7 | 50x25x2.5 | 1.9 | 17.3 | 99.4 | 4.6 | 117.7 | – | 8.8* |
| CyLS8 | 50x25x2.5 | 2.2 | 16.9 | 162.6 | 7.3 | 187.7 | – | 5.6* |
| CyLS9 | 50x25x2.5 | 2.2 | 16.9 | 165.4 | 7.0 | 190.1 | – | 5.8* |

Cyclic test programme

| | |
|---|---|
| **Cross section:** | **40x40x2.5 mm** |
| **E=** | **210000 MPa** |
| **Poisson's ratio:** | **0.3** |
| **$f_y$ =** | **343 MPa** |
| **$f_u$ =** | **393 MPa** |
| **$\varepsilon_u$ =** | **0.15** |
| **Element length (model 1)** | **1100 mm** |
| **Element length (model 2)** | **3300 mm** |

Material properties

Table 1 Test program and material properties (Goggins J.M, 2006).

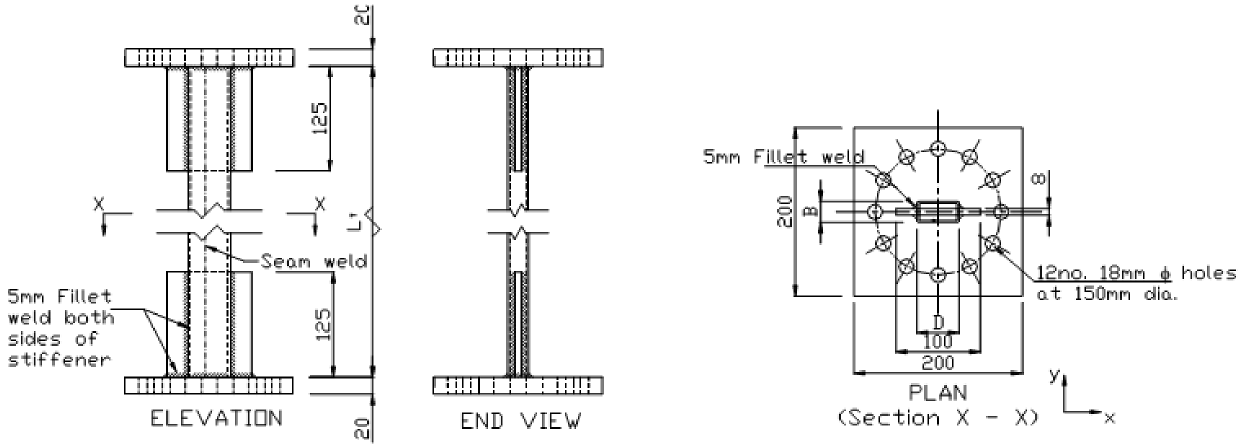



Figure 1 Test specimen (Goggins J.M, 2006)

Numerical model developed with software package (ABAQUS) [3], has the following features:

- 8-node solid elements with 6 DOF,
- elastic-plastic behavior with Von Mises Criteria,
- kinematic hardening,
- great displacement and strain.

Calculation procedure is composed of two steps:

- Buckling modes obtained from linear buckling analysis,
- Then inelastic cyclic analysis has been performed considering an initial imperfection based on first and/or second buckling mode shape.

One end of the steel strut is fully fixed, while the other end is fully fixed except for the axial degree of freedom, to which the cyclic displacement loads are applied (figure 2).

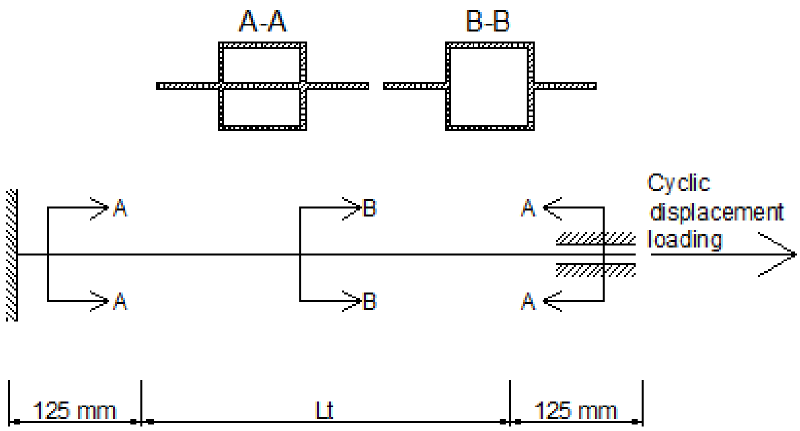



Figure 2 Setup and loading direction of the numerical model

For the element with 1100 mm length (model 1) and 3300 mm length (model 2), cyclic displacements applied with same amplitudes but with different number of cycles (Table 2).

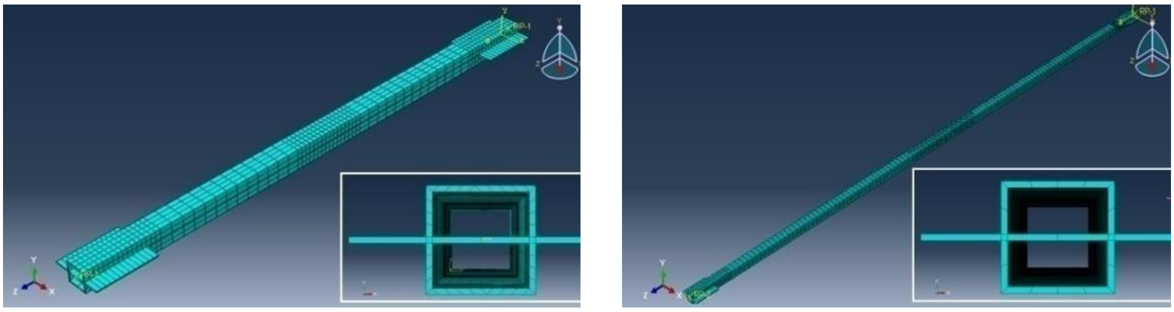
| Model 1 (1100 mm) | |
|---|---|
| one cycle | +/- 0.625 mm |
| one cycle | +/- 1.25 mm |
| one cycle | +/- 2.5 mm |
| Three cycles | +/- 5 mm |
| Three cycles | +/- 10 mm |
| Three cycles | +/- 15 mm |
| Three cycles | +/- 20 mm |
| Three cycles | +/- 25 mm |

| Model 2 (3300 mm) | |
|---|---|
| one cycle | +/- 0.625 mm |
| one cycle | +/- 1.25 mm |
| one cycle | +/- 2.5 mm |
| Three cycles | +/- 5 mm |
| Three cycles | +/- 10 mm |
| Three cycles | +/- 15 mm |
| Three cycles | +/- 20 mm |
| Three cycles | +/- 25 mm |
| Three cycles | +/- 30 mm |
| Three cycles | +/- 40 mm |

Table 2 Loading procedure (displacements are in mm)

Mesh properties are shown in figure 3. In areas where local buckling is expected, refined meshes were used (at the ends and at the center of the member). The size and the length of refined mesh areas are decided after sensitivity analysis.

Initial imperfections that are present in the structural members trigger out-of-plane deformations under forces lower than their critical buckling values. To be able to trigger this initial buckling in the inelastic cyclic analysis, initial imperfections have been incorporated in the numerical model of bracing elements. For this purpose, a linear buckling analysis has been performed, and inelastic cyclic analysis has started on the base of the deformed shapes obtained from the first and/or second linear buckling modes.

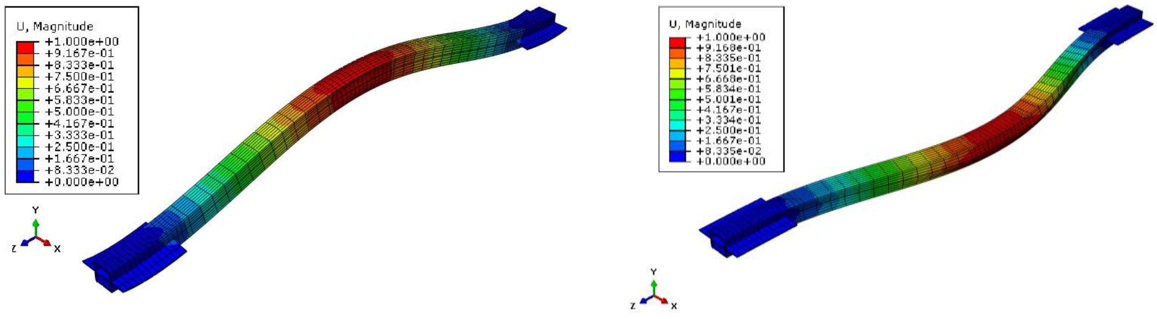


Tube 1100 mm: 2376 linear hexahedral, 444 linear wedge elements (At the mid-height and at end of the tube, for a length of 160 mm, mesh is denser).

Tube 3300 mm: 5340 linear hexahedral elements (At the mid-height and at end of the tube, for a length of 480mm and 225mm, respectively, mesh is denser).

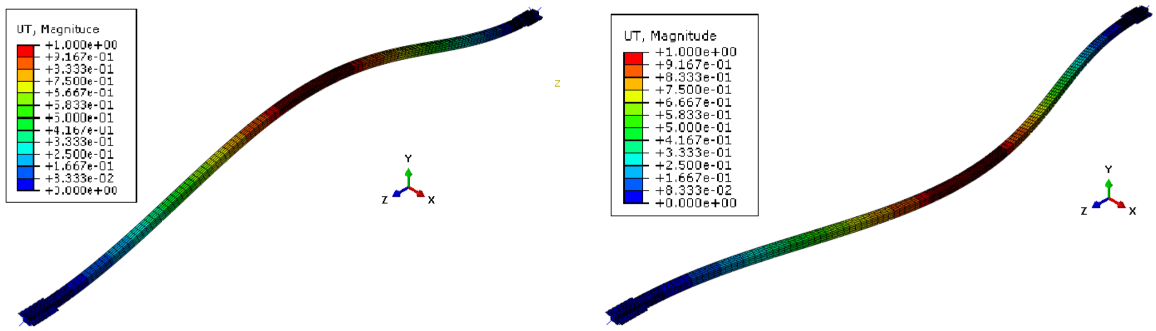
Figure 3 Mesh model 1 and model 2

In Figure 4 and Figure 5, first two linear buckling mode shapes are shown. According to the results of the experimental studies, only first mode deformed shape has been used in model 1, and both first and second mode shapes have been used for the model 2. Initial imperfection value is considered as L/150 according to Eurocode 8, however for the longer element convergence could be obtained by a larger value (L/120).




**First global buckling mode shape** **Second global buckling mode shape**

Figure 4 First and buckling mode shapes – model 1




**First global buckling mode shape** **Second global buckling mode shape**

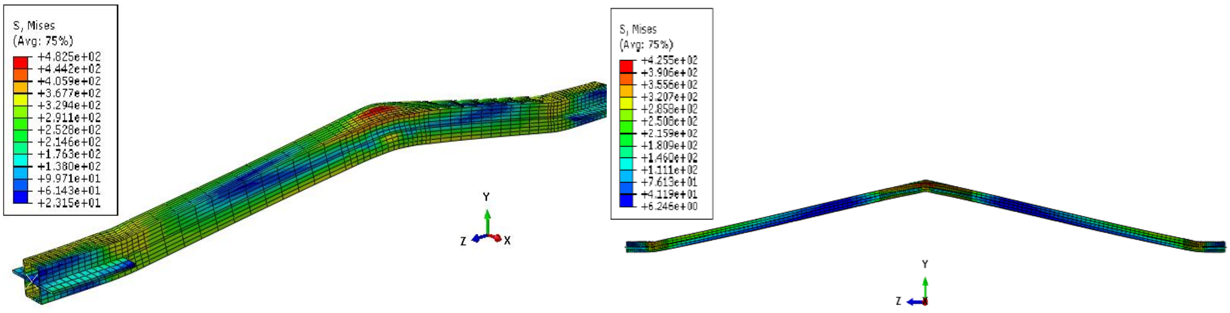
Figure 5 First and second buckling mode shapes – model 2

Critical buckling loads obtained from linear buckling analysis are almost coincident with the Euler buckling loads (Table 3).

| | | Tube 1100mm | Tube 3300mm |
|---|---|---|---|
| Critical Buckling Load | Numerical model | 573.75 KN | 66.03 KN |
| | Euler critical load | 563.20 KN | 62.58 KN |

Table 3 Comparison of critical buckling loads obtained numerically and analytically

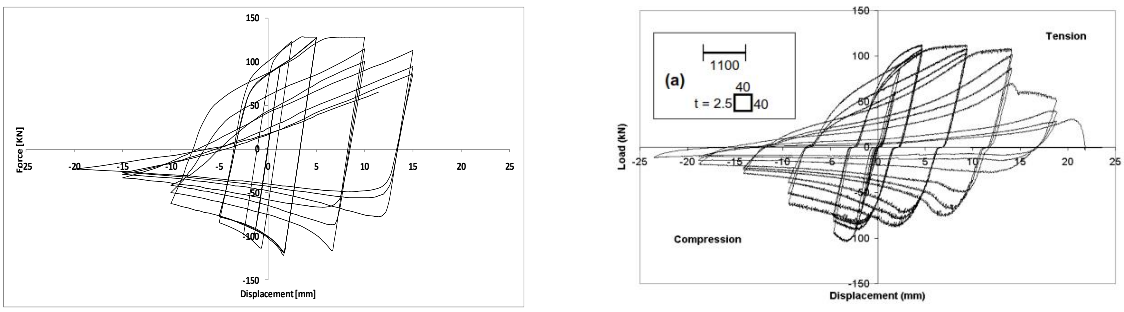
Figure 6 shows the plastic deformations concentrated at the beam ends and at the centre of the braces, as happens during the experimental tests.




**Deformed shape of tube1100 at ultimate load** **Deformed shape of tube3300 at ultimate load**

Figure 6 Deformed shape of tube elements under ultimate load

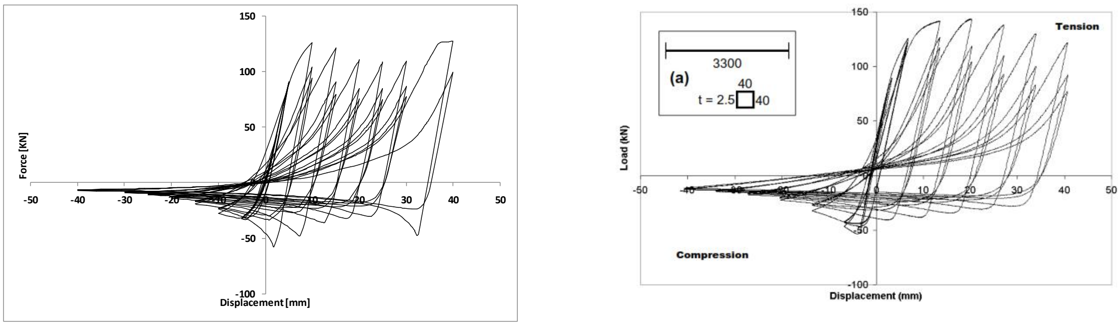
A good agreement between numerical model and experimental results has been obtained, except for the last three cycles of the displacement loading due to impossibility to reproduce local buckling fractures experienced by test specimens (figure 7). In the numerical simulation of both models, after the first buckling in compression takes place, compressive strength decreases because of plastic hinges formed at the center of the brace and next to the end plates. Then the compressive strength continues to degrade due to Baushinger effect.




a. Numerical model (Abaqus) b. Experimental result

Figure 7 Load-displacement curve - Model 1

In the case of model 2, the convergence between the experimental and numerical curve is much better, since no local buckling fracture occurred during the test (figure 8). The decrease in the global tensile and compression resistance is well visible in the numerical curve, with the exception of the last cycle. The maximum compression force recorded by the numerical model is 57.49 KN, while the experimental value is 52.30 KN. Maximum tensile force recorded by the numerical model is 127.20 KN, while the experimental value is 143.80 KN. Furthermore, in agreement with the experimental results, the numerical model provides global biaxial instability (figure 9).




a. Numerical model (Abaqus) b. Experimental result

Figure 8 Load-displacement curve - Model 2

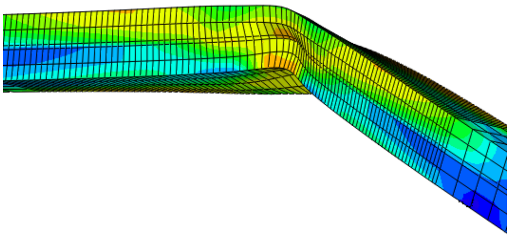


Figure 9 Biaxial instability model 2

In Table 4, comparisons are shown in terms of inelastic buckling and ultimate tensile loads according to numerical model, Eurocode 3, and experimental results. In general, it is seen that numerical simulation of more slender element gives better results.

| | | **Tube 1100mm** | **Tube 3300mm** |
|---|---|---|---|
| **Inelastic Buckling Load** | **Numerical model** | **121.66 KN** | **57.49 KN** |
| | **Eurocode** | **109.85 KN** | **43.51 KN** |
| | **Experiment** | **105.50 KN** | **52.30 KN** |
| **Ultimate tensile load** | **Numerical model** | **128.46 KN** | **127.20 KN** |
| | **Eurocode** | **147.38 KN** | **147.38 KN** |
| | **Experiment** | **112.40 KN** | **143.80 KN** |

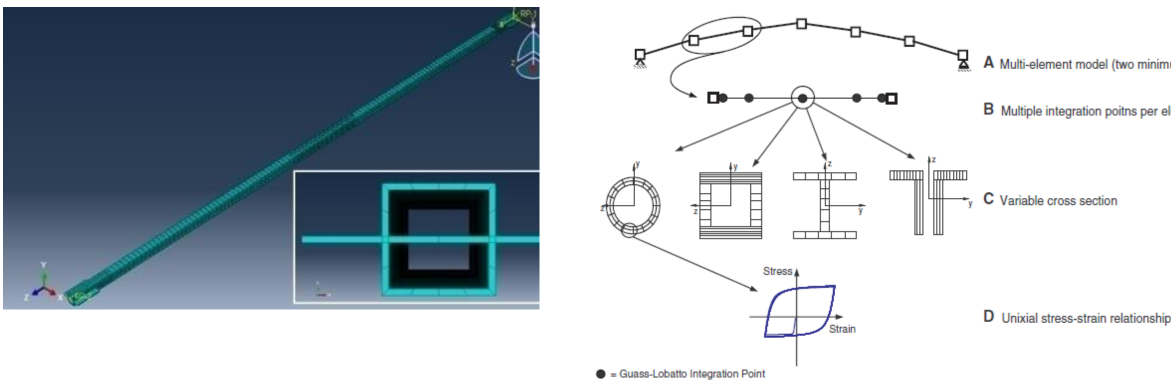
Table 4 Comparison of major loading limits between numerical analysis, experiments and code

## 2. CALIBRATION OF FIBER BASED DISTRIBUTED PLASTICITY APPROACH FOR MODELLING CBF STRUCTURES

Continuum finite element modeling (CFEM) based on shell or solid elements is currently the most efficient numerical modeling method, which is able to reproduce, with a minimum margin of error, even complex phenomena such as local buckling, distortion and changes in shape of the cross section [4]. However, this is not a common approach to study the global behavior of multistory buildings subject to seismic actions, due to its inherent complexity, computational expense, and difficulty in preparing and calibrating the models. For this reason, the CFEM method is mostly used to study the response of individual profiles or to represent details such as connection parts of global structures.

To study the global nonlinear response of multi-story buildings, most common alternatives are concentrated plasticity and fiber-based modeling approaches. Former one concentrates the inelastic deformations in individual parts of the structural system (as plastic hinges) with the rest of the structure remaining elastic. This method better captures the nonlinear response of members through calibration using test data on moment-rotation or hystereis curves. Fiber based modelling on the other hand, distributes plasticity by numerical integrations through the member cross sections and along the member length, and with a “plane sections remain plane” assumption [5]. Uniaxial material models are used to capture the nonlinear hysteretic axial stress-strain characteristics in the element cross sections. Fibers are numerically integrated over the cross section to monitor the axial force and moments, incremental moment-curvature and axial force-strain relations. The cross section parameters are numerically integrated at several sections along the member length, using displacement or force interpolation functions. This approach allows performing nonlinear analysis considering both geometric and material nonlinearity, within a time much more limited than a 3D continuum finite element analysis. However, using this approach local behaviour such as degradation due to local buckling is difficult to capture without sophisticated models. Fiber-based modeling approach with distributed plasticity (DPE) offers a good compromise in terms of accuracy and computational time to model hysteresis behavior of steel struts. Application example of this approach in moment-resisting frames is presented by Kanyilmaz et. al [6].

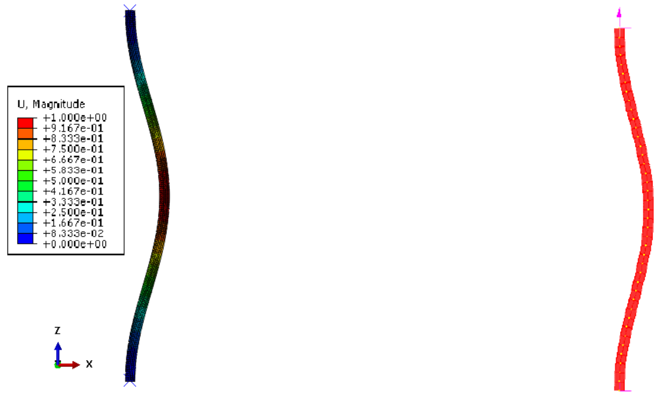
At this section, comparisons are shown between the results obtained from a CFEM model developed using Abaqus, and a fiber-based model developed using (Straus7) [7]. First, hysteresis response of the model 2 of section 1 has been compared, and then the same comparison has been made for an open section (Figure 10).




**a. CFEM approach** **b. Fiber based distributed plasticity approach [8]**

Figure 10 CFEM vs Fiber based approach

Linear buckling analysis results are very similar as can be seen in Figure 11.




**Abaqus 1° buckling mode: 66 kN** **Straus7 1° buckling mode: 67 kN**

Figure 11 Comparison in terms of 1st linear buckling mode

From the comparison of two hysteresis curves obtained from an inelastic cyclic analysis, it is evident how DPE model is able to reproduce the nonlinear response of the profile under cyclic loading (Figure 12). Both models show a gradual reduction in the maximum compression resistance at later cycles, which is in line with the actual behavior of the specimens observed during the experiments performed by [1].

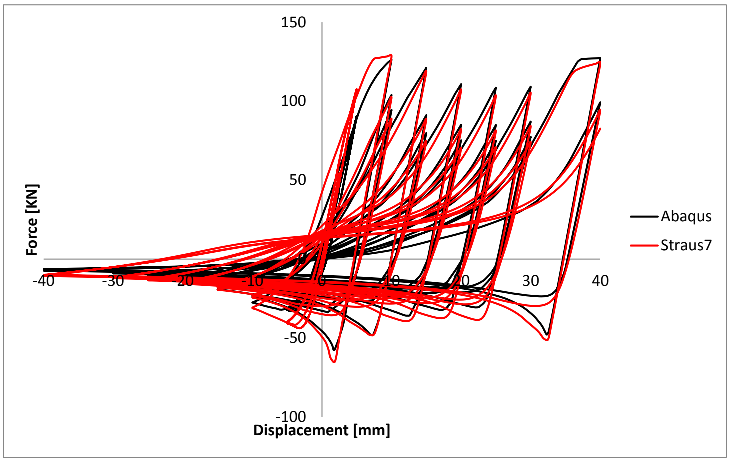



Figure 12 Comparison of hysteretic response of two modelling approaches

Maximum and minimum values achieved by the two curves are also coincident. However, in terms of dissipated energy, represented by the area enclosed by the hysteresis curves, there is a slight difference between the two models. DPE model dissipates slightly more, since DPE modeling cannot capture local instabilities that can be captured by a refined FE model with shell elements. Yet, considered the time required for the analysis and the satisfactory accuracy of DPE model, it can be concluded that the modeling with distributed plasticity (DPE) represents a good compromise between the validity of the results and analysis time.

Another comparison has been made with an open section. A DUPE100 section with 4310mm length has been analyzed. Boundary conditions are fixed on both ends for all degrees of freedom, except the axial translational degree of freedom in one end to impose the axial displacement.

Material properties are shown in Table 5. Linear buckling analysis give similar results (figure 13).

| Cross section | DUPE100 |
|---|---|
| **E: Young’s modulus** | 210000 MPa |
| **υ: Poisson coefficient** | 0,30 |
| **$f_y$: yield strength** | 343 MPa |
| **$f_u$: ultimate strength** | 393 MPa |
| **$\varepsilon_u$: ultimate strain** | 0.15 |

Table 5 Material properties


**Abaqus 1° buckling mode: 720 kN**

**Straus7 1° buckling mode: 740 kN**

Figure 13 Comparison of two approaches in terms of critical buckling load

Also in this case, results of DPE model and CFEM model are very similar. The observations made in the calibrated tube model are valid also for this case (Figure 14).

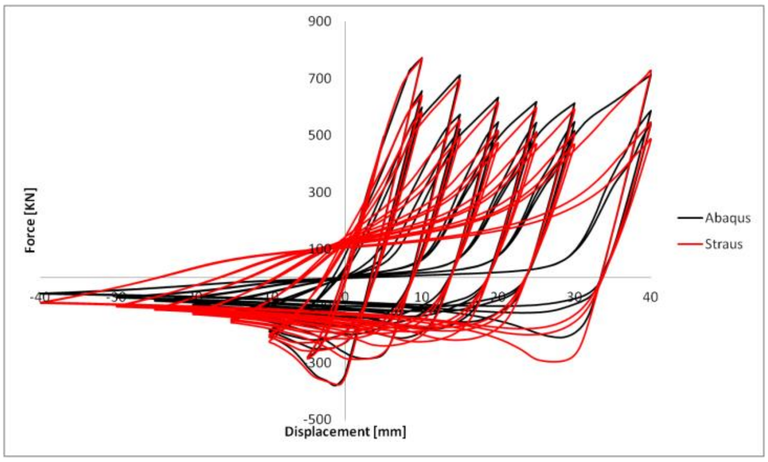



Figure 14 Comparison of hysteretic response of two modelling approaches

Local instabilities that cannot be captured by fiber-based approach are seen in Figure 15.

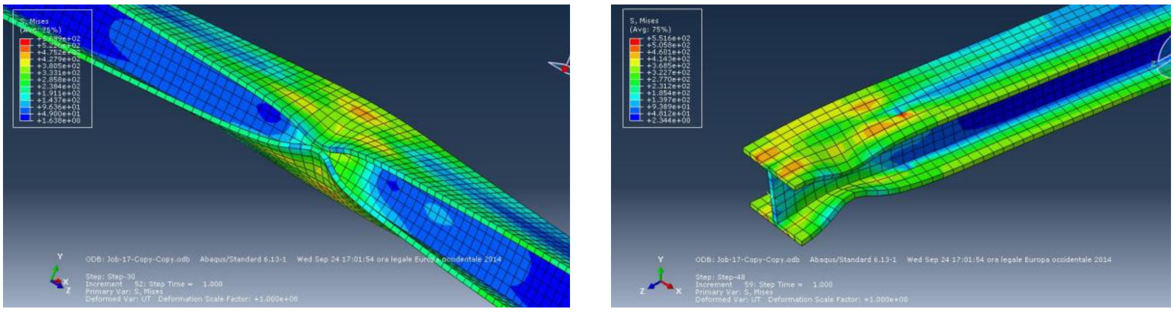


Figure 15 Local deformations at the centre and end of the strut that cannot be simulated by fiber-based approach

## 3. CONCLUSIONS

Seismic response of a concentrically braced frame mainly depends on the behaviour of its bracing elements. An accurate numerical simulation of inelastic behaviour of a braced frame is a complex matter, for which specific tools and methods are needed.

The focus of this study was to come up with an accurate and time efficient way modelling approach for the simulation of steel bracings under inelastic cyclic and seismic loading, which can be used in the modelling of multi-storey structures. Therefore, a comparative numerical study has been presented to validate the suitability of fiber based distributed plasticity modelling approach, to simulate inelastic cyclic response of bracing elements of concentrically braced frame (CBF) buildings. The simulations are based on previous experimental data and refined finite element models.

For the validation purposes, first nonlinear cyclic behaviour of various steel struts made of cold formed square hollow sections have been analyzed and calibrated with the inelastic cyclic load tests carried out by Goggins et. al. [1]. Then fiber based distributed plasticity modelling approach has been validated against calibrated continuum finite element model of steel struts.

The comparison of the hysteresis curves of several elements, evaluated for different boundary conditions shows that results obtained by fiber based approach are almost coincident with those obtained by continuum based modelling. The drawback is that fiber based modelling approach is not capable of obtaining local effects, however for a global analysis of a multi-storey building, the accuracy obtained without considering these local effects can be acceptable. The results of these analyses are used as benchmark for the simulation of braced frames within the research program MEAKADO [9,10].

## 4. ACKNOWLEDGMENT

This study has been realized thanks to the research fund received from European commission with the contract MEAKADO RFSR-CT-2013-00022.

The contribution of MSc thesis student Alex Belingheri, in performing the numerical analysis shown in this paper is deeply acknowledged.

## 5. REFERENCES

[1] J.M Goggins, B.M Broderick, A.Y Elghazouli. and A.S Lucas, Behaviour of Tubular Steel Members under Cyclic Axial Loading. *Journal of Constructional Steel Research*, **62**, 121-131, 2006

[2] ECCS (1986), Technical Committee 1. Structural Safety and Loadings -Technical Working Group 1.3. Seismic design recommended testing procedure for assessing the behaviour of structural steel elements under cyclic loads, 1st ed. Brussels

[3] ABAQUS. Analysis User's Manual. Version 6.12 ABAQUS, Inc., Dassault.

[4] S. Salawdeh, Seismic Design of Concentrically Braced Steel Frames. *PhD thesis, National University of Ireland, Galway (NUI Galway)*, 2012

[5] R. Sabelli, Research on Improving the design and analysis of earthquake resistant steel braced frames, *The 2000 NEHRP Professional Fellowship Report*, 2001

[6] A. Kanyilmaz, C.A. Castiglioni, Performance of Multi-level moment resisting composite frames with dissipative devices, *COMPDYN 2015, 5th ECCOMAS Thematic Conference on Computational Methods in Structural Dynamics and Earthquake Engineering, Crete Island, Greece, 2015*

[7] *G+D Computing Pty Limited*, Straus7. Theoretical Manual. Sydney (Australia), 2004.

[8] P. Uriz, S.A Mahin, Toward Earthquake-Resistant Design of Concentrically Braced Steel Frame Structures. *PEER Report 2008/08.* Pacific Earthquake Engineering Research Center, University of Berkeley, 2008

[9] H. Degee, C.A. Castiglioni, P.O. Martin, I. Calderon, B. Hoffmeister, Eight International Conference on Advances in Steel Structures, Lisbon, Portugal, 2015

[10] A. Kanyilmaz, C.A. Castiglioni, H. Degèe, P.O. Martin, A Preliminary assessment of slenderness and overstrength homogeneity criteria used in the design of concentrically braced steel frames in moderate seismicity, *COMPDYN 2015, 5th ECCOMAS Thematic Conference on Computational Methods in Structural Dynamics and Earthquake Engineering, Crete Island, Greece, 2015*