COMPDYN 2013
4th ECCOMAS Thematic Conference on
Computational Methods in Structural Dynamics and Earthquake Engineering
M. Papadrakakis, V. Papadopoulos, V. Plevris (eds.)
Kos Island, Greece, 12–14 June 2013

CYCLIC BEHAVIOR MODELING OF RECTANGULAR RC BRIDGE PIERS USING OPENSEES

N. Attarchian[1], A. Kalantari[1], and A.S. Moghaddam[1]

[1] International Institute of Earthquake Engineering and Seismology
No. 21, Arghavan St., North Dibajee, Farmanieh 3913/19395
Tehran Islamic Republic of Iran
{n.attarchian, a.kalantari, moghadam}@iiees.ac.ir

Keywords: Cyclic Behavior, Concrete, Bridge Pier, Fiber Modeling, OpenSees.

Abstract. *Considering various constraints in experimental program tests such as, limitations in budget, time and lab facilities, applying numerical studies in performance evaluation of structures is inevitable. While, conducting numerical analysis without adequate proficiency in modeling and analysis in nonlinear behavior range, could be deceptive. Therefore, in this paper, finite element modeling of reinforced concrete bridge piers has been established in OpenSees and, the experimental results of a couple of cyclic tests has been reproduced and compared. Fiber Modeling methodology has been elaborated and applied in this research and, the paper focuses on numerical assumptions regarding to modeling, analysis and, convergence of numerical computations. In addition the sensitivity of the local and global results to the modeling and analysis assumptions has been discussed. The numerical results obtained in OpenSees have been compared with the experimental and numerical results of previous studies in SeismoStruct program. It can be concluded that via available measures and assumptions, fiber section method has a fair capability of simulating the cyclic behavior of high and medium height piers.*

.

1 INTRODUCTION

The ideal choice for studying the actual seismic behavior of structures is, to apply full scale shaking table tests. However, due to several limitations in the laboratories, experimental tests are generally scaled and simplified to quasi static or dynamic tests. Besides, considering the time constraints and financial costs, the number of the experimental tests which can be done, even in well equipped laboratories is limited. Therefore, numerically investigating the nonlinear behavior of structures via time integration methods is inevitable. Although, this approach can expedite the analysis and, various assumptions and models can be evaluated via numerical methods, the reliability of the obtained results is very significant. Consequently, it is very essential in every numerical study to verify an experimental study which has the most resemblances and similarities with the topic under assessment.

Regarding the essence of numerical analysis in performance evaluation of structures and, the importance of verifying the results obtained from nonlinear numerical analysis, this paper focuses on the various aspects of nonlinear behavior modeling and analysis of a group of reinforced concrete (RC) bridge piers. Considering the available information about the characteristics of the specimens, loading protocols and results, cyclic behavior of piers is modeled in OpenSees [1] and, the results are compared. The studies aim to provide a reliable numerical benchmark model in order to be utilized in seismic performance evaluation of this class of RC piers.

2 SPECIMEN CHARACTERISTICS AND LOADING

The hollow rectangular RC piers under investigation are a part of a four span continuous box girder deck bridge with single column bents. The bridge is representative of typical multi-span bridges and, has been designed according to EC8 [2] provisions. The RC sections were scaled to 1:2.5 and subjected to uniaxial cyclic loading at the Joint Research Center of Ispara [3, 4]. The configuration of the bridge, the cross-sections of the deck and the reinforcement layout of piers, have been shown in Figures 1 and 2 respectively. The height of the scaled piers is 8.4 and, 5.6 m and the deck-pier connection is hinged and the lateral forces are transformed via shear keys (no transmission of moments). The longitudinal steel ratio is equal to 1.15% and the longitudinal rebars consists of $\varphi 8$, $\varphi 12$ and, $\varphi 14$ and the stirrups were $\varphi 6$ (Fig.2). The cylinder compressive strength of concrete (f_{pc}) for tall and medium height piers were 42.9 and 31.5 MPa respectively and all reinforcing bars in the piers are S500 with characteristic yield strength of 500 Mpa. Based on specific weight of concrete (25 kN/m^3), the mass per length of the deck and the column are 2.784 and, 1.664 ton/m respectively. The axial load in the piers is equal to 1700 kN and is applied at the top of the piers by means of actuators [5].

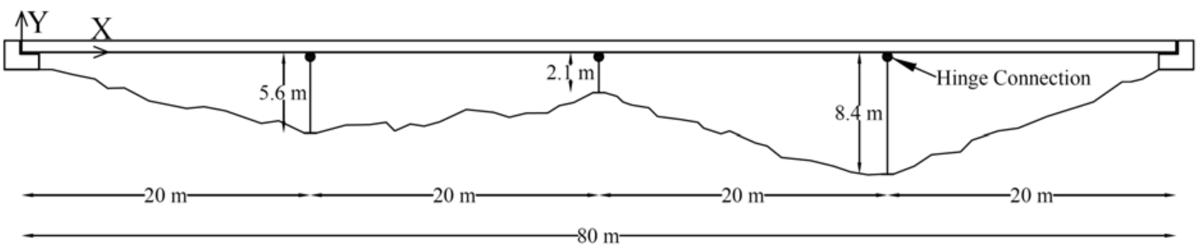


Figure 1: Bridge layout.

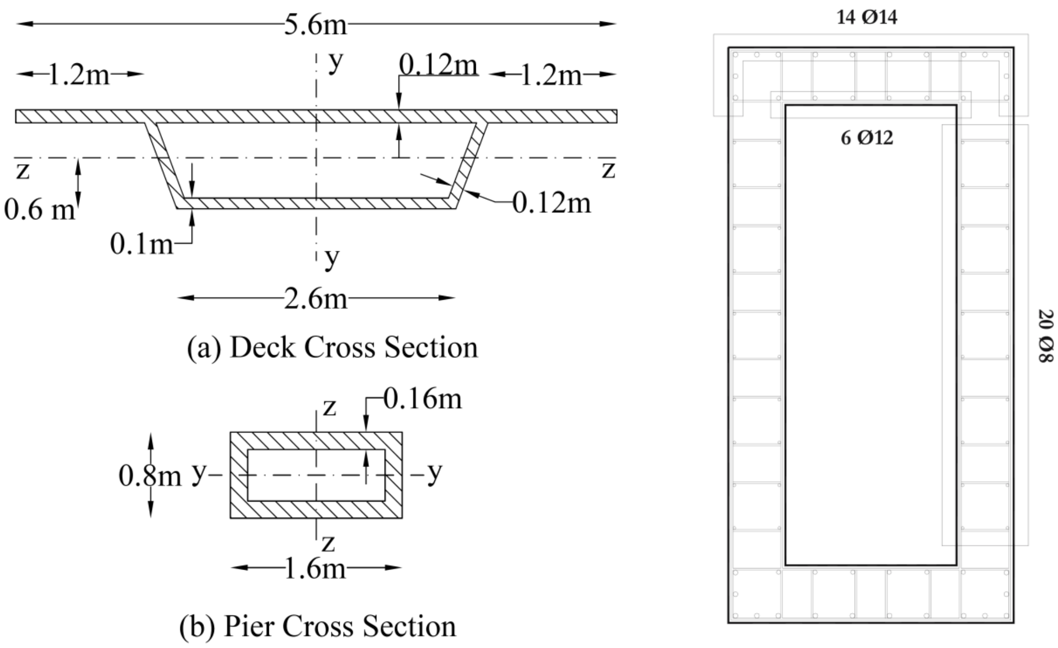


Figure 2: Pier and deck cross sections (left), Reinforcement layout of Pier Section (right).

During the cyclic test, the top of the piers is subjected to the cyclic displacement loading protocol shown in Figure 3. The tall and medium height piers were tested until failure up to 230 and 150 mm, respectively.

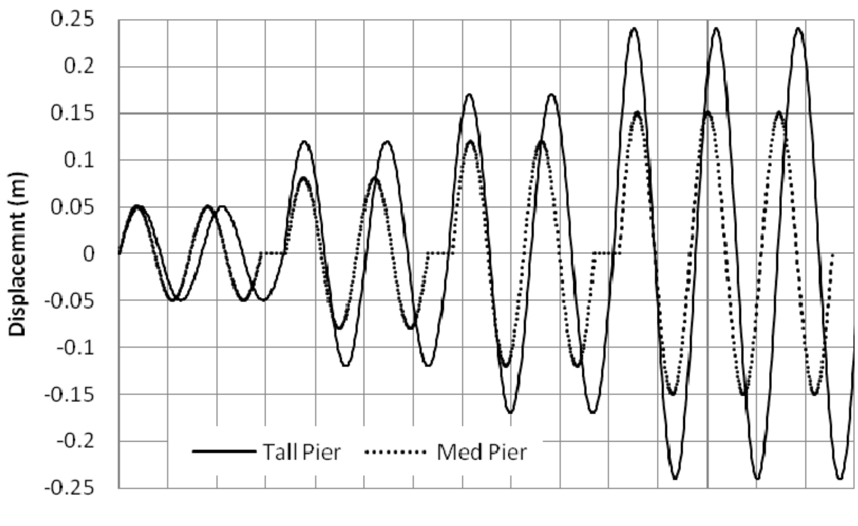


Figure 3: Cyclic test displacement histories.

3 MODELING ASPECTS

Open Source Earthquake Engineering Software (OpenSees) which, can be freely downloaded and used, enables to apply Fiber Modeling (FM) approach. Fiber modeling of frame elements is a widely practiced finite element modeling method and, can inherently account for both geometrical nonlinearities and material inelasticity, without a need for calibration of plastic hinges mechanisms typical in concentrated plasticity models [6].

In this research the piers are modeled using three dimensional beam-column elements. Since the connection of the piers to deck were hinged, piers are modeled like cantilever columns, and the tributary masses are lumped at the nodes in X and Z global direction and 1e-7 mass in assigned in Y direction (Fig. 4). Rotational inertial masses are not included.

In the case of non-rigid connections, the relative rotation of the connection can contribute to the deck drift, proportional to its vertical location. Thus, for better stimulating the deck displacement, the tributary mass of the deck is located at the height of its center of gravity, 0.6 m above the pier top, and connected by a rigid element to the pier. The rigid element was modeled via elastic beam-column element and the stiffness was set 1000 times of the deck element.

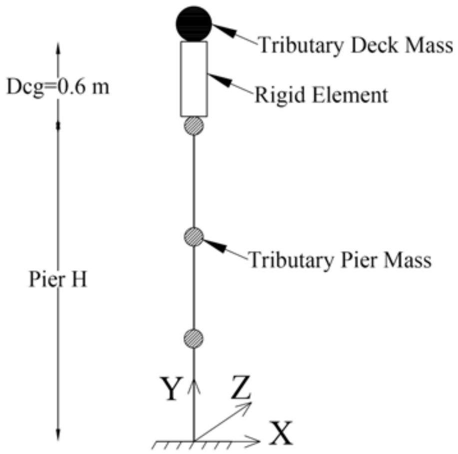


Figure 4: 3D pier modeling in OpenSees.

3.1 Material modeling

'Concrete02' uniaxial material command is used for defining confined and unconfined concrete. This model assumes linear tension softening (Fig. 5). The concrete compressive strength at 28 days is specified for the peak compression strength of unconfined concrete. Given $E_0 = 0.043W^{1.5}\sqrt{f_{pc}}$, the concrete strain at maximum strength (ε_{c0}) can be determined. The ultimate strain of unconfined concrete at crushing strength (ε_u) is specified according to Caltrans [7]. The confined concrete properties are calculated based on Mander's model [8] and the Lambda ratio between unloading slope at second step and initial slope is considered equal to 0.1 in all cases. Concrete material modeling properties for both specimens are presented in Table 1.

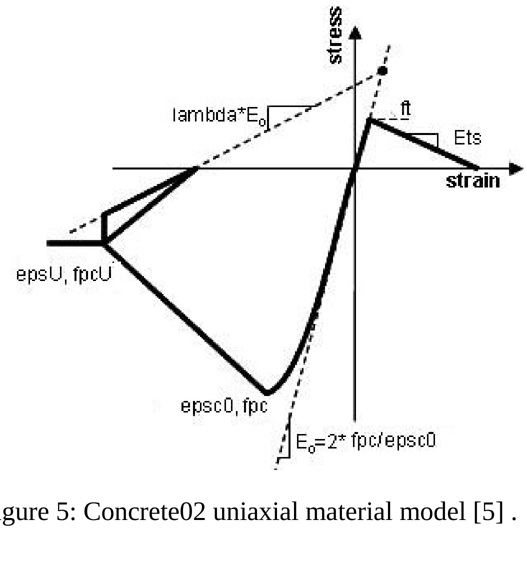


Figure 5: Concrete02 uniaxial material model [5] .

	f_{pc} (MPa)	ε_{c0}	f_{pcu} (MPa)	ε_u	f_t (MPa)	E_{ts} (GPa)
Unconfined Concrete (T)	42.9	0.003	8.6	0.005	3.1	1.05
Confined Concrete (T)	51.5	0.006	30.4	0.02	3.1	0.52
Unconfined Concrete (M)	31.5	0.0025	6.3	0.005	3.1	1.22
Confined Concrete (M)	37.8	0.005	24.2	0.03	3.1	0.6

Table 1: Concrete material modeling properties.

The applied material modeling command for steel material properties were ‘Steel02’ which is in compliance with Menegotto & Pinto constitutive law [9]. The model possesses bi-linear stress strain relationship to account for strain hardening effect (Table 2).

	F_y (MPa)	E_s (GPa)	β	R_0	C_{R1}	C_{R1}
Steel	496	203	0.0036	20	0.925	0.15

Table 2: Steel material modeling properties.

3.2 Elements modeling

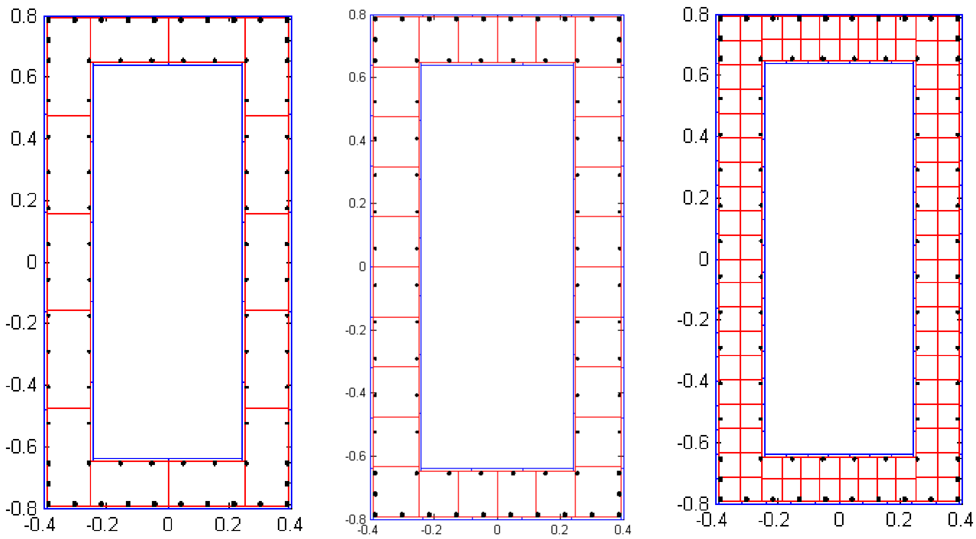
Nonlinear behavior of frame elements can be modeled via concentrated or distributed plasticity models. Although in distributed plasticity models, the interaction of moments and axial force is considered simultaneously and the plasticity is assumed to spread along the entire element length, the computation would not be very time consuming in comparison to lumped plastic hinge models, particularly in bridge systems with concentrated nonlinear behavior in piers. Considering the higher accuracy obtained via distributed plasticity models and, the simplicity of fiber modeling (omitting the need for estimating the plastic hinge length and the appropriate moment curvature relationship in lumped plastic hinges), distributed plasticity model is applied in this research.

OpenSees provides two types of distributed nonlinear beam column elements; ‘forceBeamColumn’ and ‘dispBeamColumn’. The former is based on force formulation (FBE) while the later (DBE) is based on displacement approach and follows standard finite element procedures. Since in DBE the response of the element is approximated by assuming constant axial deformation and linear curvature distribution along the element length, refine meshing of the element is needed to represent higher order distributions of deformations [10]. It should be noted that, for 3D modeling of elements in OpenSees the torsional behavior of the section should be aggregated to the fiber section element. In this regard torsion is modeled via uniaxial elastic material; the torsional rigidity of the section is considered equal to 0.2GJ according to Caltrans [7].

Since inelastic modeling can be computationally demanding, model efficiency is an important consideration. In FM approach the analysis time and the global and local responses are affected by the quantity of elements and number of integration points (NIP) along each element. Previous works [10, 11] indicate that, increasing the quantity of elements and NIPs can enhance the accuracy of global responses but may led to strain concentration and overestimation of local responses. Therefore, determining the appropriate mesh, NIPs and analysis time step which, converge to accurate responses is very significant. In this regard a parametric study is conducted to determine the efficient and accurate element type, element meshing and, NIP. According to the results, FBE provide fastest convergence with less number of elements than DBE. Each pier consists of two equal lengths, ‘forceBeamColumn’ element with 3 integration points. It is to be noted that, increasing the number of elements or NIPs would not led to more accurate results.

3.3 Section modeling

In distributed plasticity models, the cross section is segmented to fine enough fibers with proper material behavior model. In order to increase the efficiency of the model, the minimal number of fibers which could result to accurate responses, has to be specified. In this regard several options for segmenting the fiber section has been studied and the global and local responses are compared with experimental results (Fig. 6). The optimized number of fibers for the cross section is obtained in the second model with 78 fiber segments. The accuracy of the analysis would not be improved by defining more segments.



(a) 42 fiber segments (b) 78 fiber segments (c) 216 fiber segments

Figure 6: Fiber configurations under investigation.

4 ANALYSIS STAGE

Prior to cyclic analysis, gravity analysis should be done on the piers. Each type of analysis in OpenSees consists of a package of commands which will be briefly illustrated in this section. Both cyclic and gravity analysis are in the category of static analysis, and there is a great resemblance between the commands, though to expedite the computation and convergence of results, it is essential to provide compatibility, between the commands of both stages of analysis. The package of commands in each stage of analysis is summarized in Table 3. Details about the function of each of commands and other available options are elaborated further.

	Gravity analysis	Cyclic analysis
constraint	Plain	Penalty
numberer	Plain	Plain
system	BandGeneral	BandGeneral
test	EnergyIncr	EnergyIncr
algorithm	Linear	Newton
integrator	LoadControl	LoadControl
analysis	Static	Static

Table 3: Summary of analysis commands.

4.1 Constraint command

The major difference between the gravity and cyclic analysis is the constraint and algorithm commands. This command is used to construct the ConstraintHandler object. This object determines how the constraint equations are enforced in the analysis. In gravity analysis, 'Plain Constraint' provides homogeneous single point constraints can. In cyclic analysis, the displacement history at the top of the pier is defined via 'sp' command of 'pattern Plain' command and, in order to apply it in the analysis, Penalty constraint handler should be used [6].

4.2 Numberer command

In complex structures with several nonlinear elements, the numbering pattern of nodes and elements can strongly influence the bandwidth and, well planned numbering pattern can minimize the matrix bandwidth. Since in this research the model is very simple, in both stages of analysis 'Plain' numberer is applied in the analysis.

4.3 System command

There are more than six different commands in OpenSees to construct the linear system of equations and linear solver objects to store and solve the system of equations. Based on the bandwidth, population and symmetry of the system of equations, proper system command could be chosen. For efficient analysis, it is very essential to apply compatible system commands in all stages of analysis. In large and complex models, combining proper numberer command and system command could lead to more optimized solutions. Since, the structure under investigation in this research is not a large model, system 'BandGeneral' is applied in both gravity and cyclic analysis.

4.4 Convergence tests

In OpenSees the 'Test' command is used to construct a Convergence Test object over Equation (1). In all types of convergence tests in OpenSees, tolerance and maximum number of iterations have to be specified by the user. If the convergence is not achieved within the tolerance limit, more iteration will be performed. Several test types are available in OpenSees. The convergence equation for some of the options is presented in Table 4. Detailed information is available in OpenSees online manual [6]. The 'EnrgyIncr' test command, with a tolerance of 1.0e-10 and 10 maximum numbers of iterations is used in this study.

$$K\,\Delta U^{i} = R(U^{i}) \tag{1}$$

Test Command	Convergence Equation
NormUnbalance	$\lVert R(U^{i}) \rVert < tol$
test NormDispIncr	$\lVert \Delta U^{i} \rVert < tol$
EnergyIncr	$\Delta U^{i} R(U^{i}) < tol$

Table 4: Convergence equations.

4.5 Solution algorithms

Algorithm command determines the sequence of steps taken to solve the equations. Gravity analysis can be done via 'Linear' algorithms, while for non-linear behavior range 'Newton' and 'ModifiedNewton' algorithms are available. The Newton-Raphson method converges rapidly to a solution, if the initial estimate is sufficiently close to the solution, but otherwise fails to converge. In this method the tangent stiffness is updated at each iteration. In the modified Newton-Raphson method the tangent stiffness is updated only at selected steps, thus avoiding lengthy calculations needed in multi degree of-freedom systems. However, more iteration may be needed to reach a prescribed accuracy [12]. In the cyclic analysis, the nonlinear behavior of single cantilever piers under investigation has been studied via 'Newton' algorithm command. In more complicated cases, combination of both algorithms can be applied to construct the solution algorithm. So, 'ModifiedNewton' would be used as the main solution algorithm and while the convergence is not achieved after the specified maximum number of iteration, with specified tolerance; The 'Newton' algorithm would be specified as the solution algorithm.

4.6 Analysis increment and integration method

The Integrator object is used for determining the predictive step for time t+dt, specifying the tangent stiffness matrix and the residual force vector at any iteration and determining the corrective step based on the displacement increment dU. For performing static analysis, 4 types of integrators are available in OpenSees [12]. The 'LoadControl' command is used to construct a static integrator object for both stages of the analysis. The maximum load increment factor in cyclic analysis should be equal to the input steps which is equal to 0.005. The validity of the results can be verified by repeating the analysis with finer loading factor increments. Also in the cases which convergence cannot be easily obtained, finer loading increments can help to converge to the results. In this research in order to verify the results, the analysis is repeated for 0.0025 loading increment factor too. The average computational time for each of the cyclic analysis, using dual 2.00GHz processor is approximately 5 minutes.

5 ANALYSIS RESULTS

Cyclic analyses are carried out on the introduced piers under displacement loading subjected to the top of the piers. It should be noted that, beside this research, another numerical study has been done by Casarottie and Pinho [5] in SeismoStrut software [13]. For each of the piers, the base shear and displacement at the top of the piers have been derived and plotted.

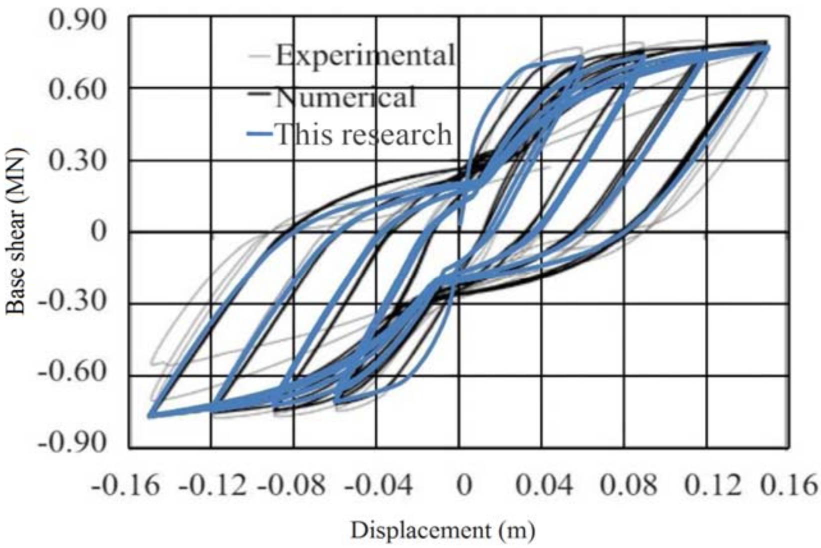


Figure 7: Cyclic experimental and numerical results for Medium Pier.

In Figures 7 and 8, the numerical results of the present study in OpenSees is compared with the previous experimental and numerical investigations, for medium and tall piers, respectively. It can be seen that, there is a very good match between the results of this study and the experimental results. Although, both numerical models cannot simulate the reduction in member strength at the very last cycle, when the failure occurs, due to the characteristic material behavior model of fibers which, would never die, the recent numerical model in OpenSees can satisfactory capture the pinching behavior in the specimens.

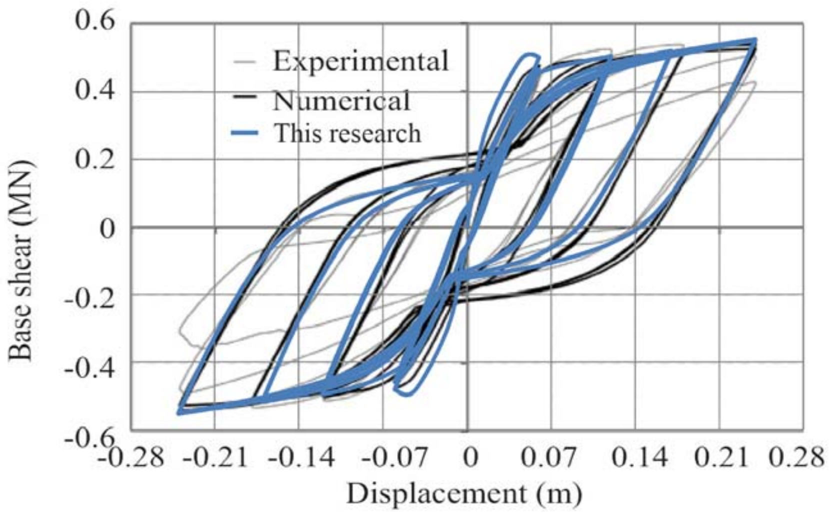


Figure 8: Cyclic experimental and numerical results for Tall Pier.

6 CONCLUSIONS

Considering the significance and necessity of numerical studies in the seismic assessment of the structures, the principals of nonlinear modeling and analysis of hollow rectangular RC bridge piers, has been elaborated in this paper and, two benchmark RC pier models have been established. The piers were modeled via nonlinear fiber section modeling approach and element properties, meshing and number of integration points of each element, also, the material behavior models and properties, have been demonstrated. Furthermore, the nonlinear analysis commands plus available and specified options have been illustrated. Results obtained in the present study indicated that the nonlinear behavior of the piers under investigation have been modeled adequately. It can be recognized that, although, due to the characteristic of material models, numerical simulations cannot capture the reduction in strength in the last cycles of test, the numerical results in the present study can finely can simulate the pinching phenomena in the hysteresis loops.

REFERENCES

[1] OpenSees Development Team, *Open System for Earthquake Engineering Simulations*, Version 2.3.0, [online]. From http://opensees.berkeley.edu, 2012.

[2] European Committee for Normalization, Eurocode 8: Design of Structures for Earthquake Resistance – Part2; Bridges, EN 1998-2: 2005.

[3] A.V. Pinto, G. Verzeletti, P. Pegon, G. Magonetto, P. Negro, J. Guedes, *Pseudo-Dynamic and shaking table tests on RC bridges* ECOEST-PrEC8, Report No.5, Brussels, 1996.

[4] G.M. Calvi, P.E Pinto, *Experimental and numerical investigations on the seismic response of bridges* ECOEST-PrEC8, Report No.4, Brussels, 1996

[5] Ch. Casarotti, R. Pinho, Seismic response of continuous span bridges through fiber-based finite element analysis *Earthquake Engineering and Engineering Vibration* **5**, 119-131, 2006.

[6] OpenSeesWiki, Online Wiki for OpenSees users and developers, Command Manual, from http://opensees.berkeley.edu/wiki/index.php/Main_Page, 2013.

[7] Caltrans Seismic Design Criteria, Version 1.6, November 2010.

[8] J.B. Mander, M.J.N. Priestle, R. Park, Theoretical Stress-strain Model for confined Concrete. *Journal of Structural Engineering, ASCE* **114**, 1804-1826, 1988.

[9] M. Menegotto, P.E. Pinto, Method of Analysis for Cyclically Loaded R.C. Plane Frames Including Changes in Geometry and Non-elastic Behavior of Elements Under Combined Normal Force and Bending. *Symposium on the Resistance and Ultimate Deformability of structures Acted on by Well Defined Repeated Loads*, International Association for Bridge and Structural Engineering, Zurich, Switzerland, September, 15-22, 1973.

[10] Discovering OpenSees, *Force-based Element vs. Displacement-based Element*, Web base semin, from:http://opensees.berkeley.edu/wiki/index.php/Discovering_OpenSees_--_Force-based_Element_vs._Displacement-based_Element, Dec 2011.

[11] M. P. Berry, M.O. Eberhard, *Performance modeling strategies for modern RC bridge columns*. PEER, Report No. 7, 2007.

[12] M. Sadrossadt Zade, M.S. Saiidi, *Pre-test analytical studies for NEESR-SG 4span bridge model using OpenSees*. CCEER, Report No. 7, 2007.

[13] SeismoStrut, SeismoSoft. *A computer program for static and dynamic nonlinear analysis of framed structures*, [online]. From http://www.seismosoft.com , 2005.