

POLITECNICO DI TORINO

Corso di Laurea Magistrale

in Ingegneria Edile

Tesi di Laurea Magistrale

Building Resilience and Sustainability Concrete Structures with the use of FRP



Relatore

prof. Gian Paolo Cimellaro

Candidato

Andrea Farinazzo

Anno Accademico 2018/2019

SUMMARY

| | |
|--|----|
| 1- ABSTRACT | 1 |
| 2- INTRODUCTION | 3 |
| 2.1- Background..... | 3 |
| 2.2- Objective of the thesis | 3 |
| 2.3- Structure of the thesis | 3 |
| 3- LITERATURE REVIEW | 4 |
| 3.1- Stress strain model for FRP confined columns..... | 7 |
| 3.2- Lam & Teng stress-strain model | 7 |
| 3.3- CNR stress-strain model..... | 10 |
| 3.4- Strengthening techniques..... | 11 |
| 3.4.1- Flexural strengthening | 11 |
| 3.4.2- Shear strengthening | 12 |
| 3.4.3- Confinement | 14 |
| 3.5- Optimization..... | 17 |
| 3.5.1- Crossover..... | 18 |
| 3.5.2- Mutation | 19 |
| 3.5.3- Genotype – phenotype mapping | 19 |
| 3.5.4- Fitness..... | 20 |
| 3.5.5- Selection | 20 |
| 3.5.6- Termination | 20 |
| 3.6- FRP and Optimization | 21 |
| 3.7- OpenSees..... | 22 |
| 4. METHODOLOGY | 22 |
| 4.1- SAP2000 elastic model building | 22 |
| 4.2- Opensees elastic model building | 23 |
| 4.3- Eigenvalue analysis | 23 |
| 4.4- Elastic response spectrum..... | 24 |
| 4.5- Uniform hazard spectrum UHS | 26 |
| 4.6- Design Response Spectrum | 26 |
| 4.7- SAP2000 non-linear model building | 28 |
| 4.8- Opensees non-linear model building | 28 |
| 4.9- Non-linear elements with plastic hinge | 29 |
| 4.10- Non-linear analysis | 31 |
| 4.11- Algorithm and integrators..... | 31 |

| | | |
|-----------|--|-----------|
| 4.12- | Non-linear static analysis..... | 31 |
| 4.13- | Non-linear dynamic analysis | 33 |
| 4.14- | Optimization strategy for FRP design | 33 |
| 5. | CASE STUDY | 36 |
| 5.1- | Materials | 36 |
| 5.2- | Geometry | 37 |
| 5.3- | Static Nonlinear Analysis | 38 |
| 5.4- | Dynamic Nonlinear Analysis..... | 40 |
| 5.5- | FINITE ELEMENT MODEL..... | 41 |
| 5.6- | Geometry | 41 |
| 5.7- | Materials | 41 |
| 5.8- | Elements | 43 |
| 6. | RESULTS | 43 |
| 6.1- | Eigenvalue analysis | 43 |
| 6.2- | Static nonlinear analysis | 45 |
| 6.3- | Dynamic nonlinear analysis..... | 45 |
| 6.4- | Optimization..... | 48 |
| 7. | CONCLUSIONS | 52 |
| 8. | APPENDIX A: OPENSEES CODE (TCL VERSION) | 53 |
| 9. | BIBLIOGRAPHY | 83 |

List of Figures

| | |
|--|----|
| Fig. 1 Stress-Strain relationship of fibres, matrix and corresponding composite..... | 5 |
| Fig. 2 FRP manual application technique..... | 6 |
| Fig. 3 Automated FRP wrapping technique | 6 |
| Fig. 4 Lam and Teng's stress-strain model for FRP-confined concrete..... | 7 |
| Fig. 5 Effective confinement for rectangular sections | 9 |
| Fig. 6 Stress-strain model for FRP confined concrete..... | 10 |
| Fig. 7 Typical flexural strengthening of RC beams..... | 11 |
| Fig. 8 Dependence of FRP elastic modulus on fibre orientation..... | 13 |
| Fig. 9 Shear wrapping schemes | 13 |
| Fig. 10 Stress-strain behavior of unconfined and confined RC columns (Rocca et al.2006)..... | 14 |
| Fig. 11 Discontinuous confinement of a circular section..... | 16 |
| Fig. 12 Confinement of a rectangular section..... | 16 |
| Fig. 13 Opensees Script example | 22 |
| Fig. 14 Response Spectrum construction | 24 |
| Fig. 15 Acceleration, velocity and displacement response spectra for Kozani-prefecture record..... | 25 |
| Fig. 16 Hazard Deaggregation for the city of Rome | 26 |
| Fig. 17 Design Spectrum typical shape | 27 |
| Fig. 18 Modified Gauss-Radau plastic hinge integration methods..... | 30 |
| Fig. 19 Capacity curve..... | 32 |
| Fig. 20 Five story RC frame views: a) 3D view; b) 2D view..... | 37 |
| Fig. 21 Beam and column cross-section..... | 37 |
| Fig. 22 Elements ID number | 38 |
| Fig. 23 Elastic Design Spectrum | 39 |
| Fig. 24 Scaled Design Spectrum..... | 39 |
| Fig. 25 Concrete02 stress-strain relationship - opensees.berkeley.edu | 42 |
| Fig. 26 Steel02 stress-strain relationship - Opensees.berkeley.edu..... | 42 |
| Fig. 27 Stress-strain relationship for square section with and without FRP - (McKenna 2011) | 43 |
| Fig. 28 Comparison of modal periods for Elastic frame | 44 |
| Fig. 29 Fundamental periods comparison..... | 44 |
| Fig. 30 Capacity curve for five story RC frame | 45 |
| Fig. 31 Displacement comparison between SAP2000 and OpenSees for direction 1 | 46 |
| Fig. 32 Displacement comparison between SAP2000 and OpenSees for direction 3 | 46 |
| Fig. 33 Interstory Drift ratio – dir. 1..... | 47 |
| Fig. 34 Interstory Drift ratio – dir. 3..... | 47 |
| Fig. 35 Displacements comparison dir. 1 | 48 |
| Fig. 36 Interstory drift ratio for optimal solution – dir.1 | 49 |
| Fig. 37 Interstory drift ratio for optimal solution – dir.3 | 49 |
| Fig. 38 GA solutions..... | 49 |
| Fig. 39 Admissible solutions | 50 |
| Fig. 40 Probability of columns of being confined with FRP during the optimization algorithm | 50 |
| Fig. 41 Optimal disposition of FRP confined columns | 51 |
| Fig. 42 Interstory drift ratio for columns most interested by FRP wrapping..... | 51 |

List of Tables

| | |
|--|----|
| Table 1 Properties of matrix and fibers | 5 |
| Table 2 Generic GA workflow | 18 |
| Table 3 Calculation of effective confinement c_{eff} | 38 |
| Table 4 Determination of static forces | 40 |
| Table 5 Interstory drift ratio | 47 |

1- ABSTRACT

The seismic retrofit topic is very debated nowadays, especially for those Countries that can count on a large number of existing, old and historical buildings. Many of these cannot be repaired or retrofitted through the most common systems (i.e. shear walls, base isolators, dampers...) due to architectural constraints and/or to budget limitations. Thus, the Fiber Reinforced Polymers (FRP) impose themselves as one of the best seismic retrofit system for existing buildings. The International Codes provide standards concerning the design of FRP wraps for a certain geometry and for a certain stress, but a set of rules for the placement on the structural elements is unknown.

Therefore, this thesis aims to find the optimal layout of FRP-confined columns within a five story reinforced concrete frame, subjected to several nonlinear time history analysis. The inelastic numerical model has been realized using OpenSees (McKenna 2011). The optimization procedure is based on the minimization of the objective function C , defined as the square root of the sum of the squares of the interstory-drift ratios.

With the aim of make more performing the procedure, the optimization research has been based on the Genetic Algorithms, that are biology-inspired methods used for global optimization, that seek the solution in the form of string of numbers, evolving at each iteration.

The results have shown that there is a correlation more evident between the position of FRP-wrapped column and the minimization of C rather than the minimization of C and the number of FRP-wrapped columns. In particular the analysis have highlighted that is better to retrofit those columns where the maximum interstory drift ratio is expected.

Key words: FRP, optimization, Genetic Algorithms, Reinforced Concrete frame, OpenSees

2- INTRODUCTION

2.1- Background

The issue of strengthening and retrofitting existing civil infrastructure is very debated nowadays. When dealing with existing buildings any engineer is aware of the troubles due to the architectural constraints or how important is to limit the aesthetic impact. These reasons helped the externally bonded FRP systems to impose themselves as one of the best seismic retrofit system for existing infrastructures during the last three decades. FRP is the acronym of Fiber Reinforced Polymers, that are composite materials made by a polymeric matrix and by fibers of three different materials: they can be realized in carbon, glass or aramid. These composite materials are heterogeneous and anisotropic, and they predominantly have an elastic behaviour up to failure. The reasons why these materials are widely used as retrofitting and strengthening materials for Reinforced Concrete buildings are summarised as follow: low weight, high tensile strength, large deformation capacity, immunity to corrosion and unlimited availability in sizes, dimensions and geometry. They also suffer from certain disadvantages, the most significant concerns their behaviour: contrary to steel, which as an elastoplastic behaviour, composites in general are linear elastic to failure, without plastic deformation and with a reduced ductility. Moreover, FRP materials are very sensitive to high temperatures that can cause premature degradation.

2.2- Objective of the thesis

The aim of this Master Thesis is to assess the performance of a five story reinforced concrete frame, who wants to simulate the non-ductile behavior of buildings realized before the latest Seismic Standards (i.e. in the second half of 1900). Once that the need of strengthening the building is ascertained, a retrofit system is proposed: in particular the retrofit through FRP confinement is studied. The ultimate goal is to find an optimal solution for the arrangement of the FRP-confined columns among the twenty columns available. To achieve this purpose a numerical model of the frame is realized using the Open System for Earthquake Engineering Simulation (OpenSees) (McKenna 2011), a structural analysis software developed at PEER by Frank McKenna and Gregory L. Fenves. This numerical model is the basis around which the optimization algorithm, used to find the optimal solution, is developed.

2.3- Structure of the thesis

In the first part of the Thesis is presented the theme of the Fiber Reinforced Polymers materials with their behavior and their application and also the optimization algorithm used are presented. The second portion of the Thesis is focused on the methodology used, with the description of the building of the model and the analysis performed. In the third part is presented the case study while in the fourth part the results are shown.

3- LITERATURE REVIEW

FRP systems are available in a variety of forms, including monodirectional pultruded sheet and bidirectional fabric. They can be grouped in three different classes, due to their morphology: *wet layup* systems, *prepreg* systems and *precured* systems. According with (ACI-Committee-440 2008) “*Wet layup* FRP systems consist of dry unidirectional or multidirectional fiber sheets or fabrics impregnated with a saturating resin on site. The saturating resin, along with the compatible primer and putty, bonds the FRP sheets to the concrete surface. Wet layup systems are saturated in place and cured in place and, in this sense, are analogous to cast-in-place concrete. *Prepreg* FRP systems consist of partially cured unidirectional or multidirectional fiber sheets or fabrics that are pre-impregnated with a saturating resin in the manufacturer’s facility. Prepreg systems are bonded to the concrete surface with or without an additional resin application, depending on specific system requirements. Prepreg systems are saturated off-site and, like wet layup systems, cured in place. *Precured* FRP systems consist of a wide variety of composite shapes manufactured off site. Typically, an adhesive, along with the primer and putty, is used to bond the precured shapes to the concrete surface. The system manufacturer should be consulted for recommended installation procedures. Precured systems are analogous to precast concrete.”

The two main phases that compose these materials are the matrix and the fibers. In general, the matrix for a structural composite material can be of two different types: thermosetting or thermoplastic. The most common one is the epoxy resins which are part of the first matrix class. According with (fib 2001) “the matrix has the function to protect the fibres, against corrosion or abrasion, to distribute the load and to bind the fibres together”. Several mechanical properties of the composite are influenced by the matrix, such as the shear and the compression properties.

The load-bearing function is performed by the fibers, that can be manufactured in continuous or discontinuous form and that can be classified in three different types based on the constituent material: glass, carbon and aramid. Thanks to their properties, the carbon fibers are the most used in the civil field, in fact they are cheaper than glass or aramid and they show a higher elastic modulus and tensile strength.

| | Elastic Modulus E | Tensile strength σ | Ultimate strain ϵ | Thermal Expansion Coefficient α | Density ρ |
|---------------------------------|------------------------------------|---|--|--|--|
| | [GPa] | [MPa] | [%] | [$10^{-6} \text{ } ^\circ\text{C}^{-1}$] | [g/cm ³] |
| Glass fibre E | 70-80 | 2000-3500 | 3.5-4.5 | 5-5.4 | 2.5-2.6 |
| Glass fibre S | 85-90 | 3500-4800 | 4.5-5.5 | 1.6-2.9 | 2.46-2.49 |
| Carbon fibre (High Modulus) | 390-760 | 2400-3400 | 0.5-0.8 | -1.45 | 1.85-1.9 |
| Carbon fibre (High Strength) | 240-280 | 4100-5100 | 1.6-1.73 | -0.6 - -0.9 | 1.75 |
| Aramid fibre | 62-180 | 3600-3800 | 1.9-5.5 | -2 | 1.44-1.47 |
| Polymeric matrix | 2.7-3.6 | 40-82 | 1.4-5.2 | 30-54 | 1.10-1.25 |

Table 1 Properties of matrix and fibers

Therefore, Fiber Reinforced Polymers consist of the union of non-metallic fibres bundled in a resin matrix. Following the definition given by (fib 2001) these composite materials are distinguished according “on the type of fibre they are referred to as AFRP (aramid fibre based), CFRP (carbon fibre based) or GFRP (glass fibre based).” For a unidirectional prepreg system the elastic modulus and the tensile strength are lower than those of fibers, while the ultimate tensile strain is comparable, as shown in Fig. 1.

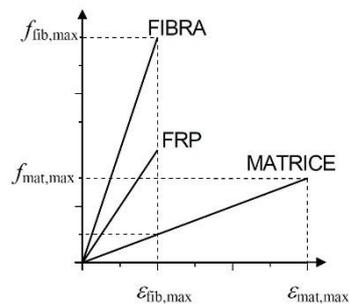


Fig. 1 Stress-Strain relationship of fibres, matrix and corresponding composite

According with (ACI-Committee-440 2008) “when loaded in direct tension, unidirectional FRP materials do not exhibit any plastic behavior (yielding) before rupture. The tensile behavior of FRP materials consisting of one type of fiber material is characterized by a linear elastic stress-strain relationship until failure, which is sudden and brittle.

The tensile strength and stiffness of an FRP material is dependent on several factors. Because the fibers in an FRP material are the main load-carrying constituents, the type of fiber, the orientation of fibers, the quantity of fibers, and method and conditions in which the composite is produced affect the tensile properties of the FRP material.” While the characteristics of FRP in tension are known they cannot be used as “compression reinforcement due to insufficient testing validating its use in this type of application.”

Some of the most popular techniques for upgrading RC elements traditionally involves the use of steel in form of “plates epoxy bonded to the external surfaces of beams and slabs” or in form of jackets around RC elements, that can also be realized in concrete. The first technique in particular, is simple and effective, but suffers of some disadvantages, such as the heavy weight and the corrosion, that also affects the steel jackets. “Replacing the steel plates and the steel jackets with FRP strips or FRP sheets provides satisfactory solutions to the problems described above.”

In general, FRP strengthening technique involves the manual application (Fig. 2) of the systems listed before or the automated wrapping. In the first case, “the external reinforcement is bonded onto the concrete surface with the fibres as parallel as practically possible to the direction of principal tensile stress.”



Fig. 2 FRP manual application technique

“The automated FRP wrapping technique (Fig. 3) was first developed in Japan in 90s and it involves continuous winding of wet fibres under a slight angle around columns or other structures by means of a robot”. This technique allows a rapid installation of the composite.



Fig. 3 Automated FRP wrapping technique

The adherence between composite and concrete has a great importance when dealing of upgrading intervention because the failure for debonding of the adhesive between the substrate and the strengthening system is brittle.

For the hierarchy of resistance, this failure type cannot precede the flexural or shear collapse of the reinforced element.

3.1- Stress strain model for FRP confined columns

From the end of 90s and the beginning of 00s various models for the behavior of RC section confined with FRP were produced. All these models found themselves on the well-established fact that wrapping a RC section with FRP sheets produce an enhancing in axial strength and in ductility when the elements are subjected to axial loads. Many of these models were produced after several tests on circular columns, where the confinement effect is uniformly distributed along the cross section, while for rectangular cross-section even the production of tests was limited. Thanks to (Lam and Teng 2003) a stress-strain model for rectangular sections was proposed: it was basically, a modification of their previous work on circular columns.

3.2- Lam & Teng stress-strain model

The model proposed from Lam and Teng (2003) comes directly from the formulation of the stress-strain relationship for uniformly confined section, advanced by the same authors. “This model is based on the following assumptions: (i) the stress–strain curve consists of a parabolic first portion and a straight-line second portion, as shown in Fig. 4; (ii) the initial slope of the parabola is the same as the elastic modulus of unconfined concrete E_c ; (iii) the nonlinear part of the first portion is affected to some degree by the presence of an FRPjacket ; (iv) the parabolic first portion meets the linear second portion smoothly (i.e. there is no change in slope between the two portions where they meet); (v) the linear second portion ends at a point where both the compressive strength f'_{co} and the ultimate axial strain ϵ_{cu} of confined concrete are reached.” (Lam and Teng 2003)

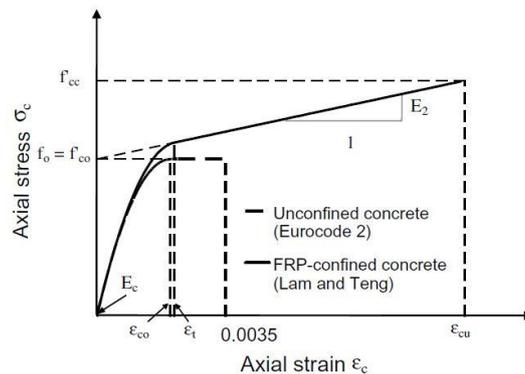


Fig. 4 Lam and Teng's stress-strain model for FRP-confined concrete

These assumptions are provided and verified for a confinement ratio (eq.(10)) bigger than 0.07. Moreover, the intercept of the stress axis by the linear second portion f_0 is taken to be equal to the unconfined compressive strength f'_{co} . The stress-strain model is then given by:

$$\sigma_c = \begin{cases} E_c \varepsilon_c - \frac{(E_c - E_2)^2}{4f'_{co}} \varepsilon_c^2 & (0 \leq \varepsilon_c \leq \varepsilon_t) \\ f'_{co} + E_2 \varepsilon_c & (\varepsilon_t \leq \varepsilon_c \leq \varepsilon_{cu}) \end{cases} \quad (1)$$

Where

$$E_2 = \frac{f'_{cc} - f'_{co}}{\varepsilon_{cu}} \quad (2)$$

$$\varepsilon_t = \frac{2f'_{co}}{(E_c - E_2)} \quad (3)$$

In eq.(1) σ_c and ε_c represent the compressive stress and compressive strain respectively of confined concrete; E_c is the initial elastic modulus, f'_{co} is the compressive strength of unconfined concrete, E_2 is the slope of the straight line that intercepts the stress axis at $f_0 = f'_{co}$, ε_t represent the strain in which the parabolic portion meets the linear portion. In eqs.(2) and (3) f'_{cc} is the compressive strength of FRP confined concrete, while ε_{cu} is the corresponding ultimate compressive strain. Both quantities can be obtained respectively from:

$$\frac{f'_{cc}}{f'_{co}} = 1 + 3.3k_{s1} \frac{f_l}{f'_{co}} \quad (4)$$

$$\frac{\varepsilon_{cu}}{\varepsilon_{co}} = 1.75 + 12k_{s2} \left(\frac{f_l}{f'_{co}} \right) \left(\frac{\varepsilon_{h,rupt}}{\varepsilon_{co}} \right) \quad (5)$$

As mentioned by (Lam and Teng 2003) “in eqs. (4) and (5), ε_{co} is the axial strain at peak stress of unconfined concrete, taken as 0.002; $\varepsilon_{h,rupt}$ is the FRP hoop rupture strain; f is the equivalent confining pressure, defined as follows; and k_{s1} and k_{s2} are the shape factors for strength and ultimate strain, respectively”.

The eqs. (4) and (5) are the same provided for circular sections but differ from them for the introduction of two shape factors, k_{s1} and k_{s2} respectively. These two factors are introduced to take into account of the non-uniform confinement provided by FRP wraps for rectangular sections, as shown in Fig. 5

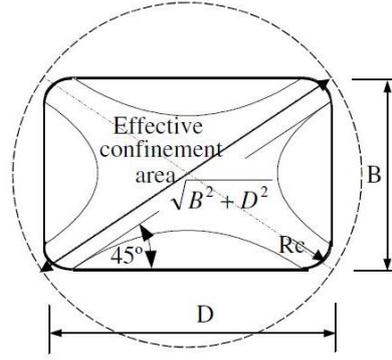


Fig. 5 Effective confinement for rectangular sections

The two factors are obtained from a geometrical construction, that consider an equivalent wrapped circular section with the diameter equal to the bigger dimension of a rectangular section: thus, the equivalent confining pressure is the one provided by an FRP wraps with the same thickness. Thus, the confining pressure is:

$$f_l = \frac{2E_{frp}t\varepsilon_{h,rup}}{\sqrt{D^2 + B^2}} \quad (6)$$

Where E_{frp} is the elastic modulus of FRP. Instead, the two shape factors are:

$$k_{s1} = \left(\frac{B}{D}\right)^2 \frac{A_e}{A_c} \quad (7)$$

$$k_{s2} = \left(\frac{D}{B}\right)^{0.5} \frac{A_e}{A_c} \quad (8)$$

Following (Lam and Teng 2003) “ A_e is the area of the effectively-confined concrete and A_c is the total area of concrete enclosed by the FRP jacket. The effective confinement area ratio is given by:

$$\frac{A_e}{A_c} = \frac{1 - \frac{\frac{B}{D}(D - 2R_c)^2 + \frac{D}{B}(B - 2R_c)^2}{3A_g} - \rho_{sc}}{1 - \rho_{sc}} \quad (9)$$

Where A_g is the gross area of the section; ρ_{sc} is the cross-sectional area ratio of longitudinal steel; and R_c is the radius of the rounded corners”. Smooth the corner of a rectangular section is a recommended procedure to improve the effectiveness of confinement.

The authors provide also a simple relation, expressed by the following eq.(10) , to estimate the effectiveness of the confinement:

$$c_{eff} = \frac{f_l}{f_{co}} \quad (10)$$

The model presented by Lam and Teng is all in all simple and well describes the behaviour of the FRP-confined concrete. Furthermore, it allows to reduce directly to the stress-strain relationship, provided for example by the Eurocode 2, for unconfined concrete.

3.3- CNR stress-strain model

The Italian NTC2018 does not provide directly a model for FRP-confined sections. This task was delegated to the *Consiglio Nazionale delle Ricerche* (CNR 2014), who provided a stress-strain model similar to that of Lam and Teng for axial compression stress condition. It consists of a parabolic portion and an increasing linear portion (Fig. 6).

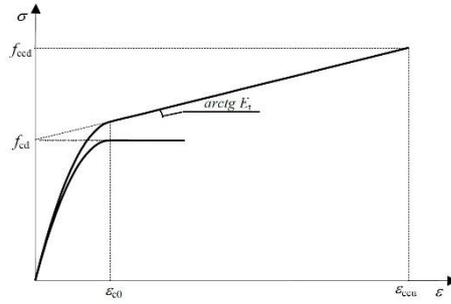


Fig. 6 Stress-strain model for FRP confined concrete

This model is analytically described by the followings:

$$\frac{f_c}{f_{cd}} = 1 + a \cdot \bar{\varepsilon} - \bar{\varepsilon}^2 \quad \text{per} \quad 0 \leq \bar{\varepsilon} \leq 1 \quad (11)$$

$$\frac{f_c}{f_{cd}} = 1 + b \cdot \bar{\varepsilon} \quad \text{per} \quad 1 \leq \bar{\varepsilon} \leq \frac{\varepsilon_{ccu}}{\varepsilon_{c0}} \quad (12)$$

Where: $\bar{\varepsilon} = \frac{\varepsilon_c}{\varepsilon_{c0}}$ is a dimensionless coefficient, f_{cd} and ε_{c0} are, respectively, the compressive unconfined concrete strength and the corresponding strain (generally equal to 0.002), ε_{ccu} is the ultimate strain of confined concrete, the two coefficients a and b are $a = 1 + \gamma$ and $b = \gamma - 1$. Furthermore:

$$\gamma = \frac{f_{cd} + E_t \cdot \varepsilon_{c0}}{f_{cd}} \quad (13)$$

$$E_t = \frac{f_{ccd} - f_{cd}}{\varepsilon_{ccu}} \quad (14)$$

Where f_{ced} is the ultimate compressive strength used for design. This quantity (i.e. f_{ced}) will be presented below in the chapter concerning the confinement, where the stress-strain model for rectangular FRP-confined section provided by CNR will be discussed.

3.4- Strengthening techniques

The use of these composite materials becomes an effective solution when a civil infrastructure needs to be upgraded. Recently, the (fib 2001) bulletin has reported that FRP have been used for “strengthening reinforced concrete (RC) elements (such as beams, slabs, columns etc.) in thousands of applications worldwide, where conventional strengthening techniques may be problematic, such as: deterioration attributed to ageing, environmentally deterioration, lack of maintenance and accidental events such as earthquakes”.

In the next chapters the most commonly used strengthening techniques are presented.

3.4.1- Flexural strengthening

FRP composite materials can be used for flexural strengthening, in fact (ACI-Committee-440 2008) suggested that “bonding FRP reinforcement to the tension face of a concrete flexural member with fibers oriented along the length of the member will provide an increase in flexural strength”. This retrofit system can involve wet lay up systems or precured systems. Experimental evidence suggests wrapping each RC element with a maximum of five plies of FRP sheets (for a thickness of 1.0-1.5 mm), to ensure the effectiveness of the strengthening system.

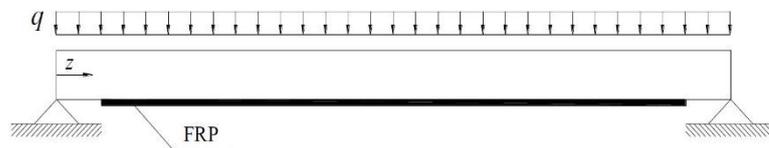


Fig. 7 Typical flexural strengthening of RC beams

The Ultimate Limit State design requires that the design flexural strength, referring to the nominal strength of the member, be greater than the design bending stress. The strengthening intervention is particularly suggested for those elements with a weak longitudinal steel reinforcement. According with (fib 2001) “the analysis for the ULS in flexure for such elements may follow well-established procedures for reinforced concrete structures, provided that: the contribution of external FRP reinforcement is taken into account properly; and special consideration is given to the issue of bond between the concrete and the FRP”. The (ACI-Committee-440 2008) described that “the nominal flexural strength of FRP-strengthened concrete members with mild steel reinforcement and with bonded prestressing steel can be determined based on strain compatibility, internal force equilibrium, and the controlling mode of failure. The flexural strength of a section depends on the controlling failure mode. The following modes should be investigated for an FRP-strengthened section:

- Crushing of the concrete in compression before yielding of the reinforcing steel;

- Yielding of the steel in tension followed by rupture of the FRP laminate;
- Yielding of the steel in tension followed by concrete crushing;
- Shear/tension delamination of the concrete cover (cover delamination);
- Debonding of the FRP from the concrete substrate (FRP debonding).”

It is also significant, in case of flexural strengthening of a RC element, to verify that “the member should be capable of resisting the shear forces associated with the increased flexural strength. The potential for shear failure of the section should be considered by comparing the design shear strength of the section to the required shear strength. If additional shear strength is required, FRP laminates oriented transverse to the beam longitudinal axis can be used to resist shear forces”.

Due to the elastic behaviour of FRP up to failure, “the level of strain in the FRP will dictate the level of stress developed in the FRP. The maximum strain level that can be achieved in the FRP reinforcement will be governed by either the strain level developed in the FRP at the point at which concrete crushes, the point at which the FRP ruptures, or the point at which the FRP debonds from the substrate. The effective strain level in the FRP reinforcement at the ultimate limit state can be found from:”

$$\varepsilon_{fe} = \varepsilon_{cu} \left(\frac{d_f - c}{c} \right) - \varepsilon_{bi} \leq \varepsilon_{fd} \quad (15)$$

Moreover, the application of externally bonded FRP reinforcement for flexural strengthening can cause a reduction of the ductility of the original element. The loss in ductility is in most cases negligible, however, to ensure a certain degree of ductility the strain “level in the steel at the ultimate limit state should be checked”, as described in (ACI-Committee-440 2008).

3.4.2- Shear strengthening

FRP are also widely used for shear strengthening of reinforced concrete elements. In fact, (ACI-Committee-440 2008) defined that “the additional shear strength that can be provided by the FRP system is based on many factors, including geometry of the beam or column, wrapping scheme, and existing concrete strength”. To maximise the effectiveness of the composite material, FRP sheets are placed on the external side “with the principal fibre direction as parallel as possible to that of maximum principal tensile stress”, as declared by the (fib 2001) report. Indeed, the FRP elastic modulus has a great dependence on the fibre orientation (Fig. 8).

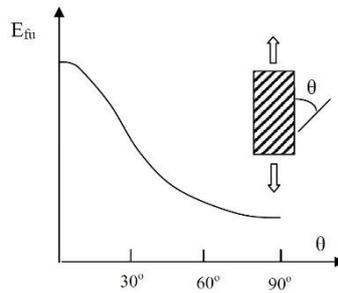


Fig. 8 Dependence of FRP elastic modulus on fibre orientation

In the Civil Engineer field, in which the structures are particularly affected by loads perpendicular to the element axis (e.g. seismic load for columns or gravity load for beams), the principal stress trajectories form an angle with the member axis usually taken equal to 45° or 90° .

As mentioned above, the effectiveness of FRP shear strengthening is conditioned also by the wrapping scheme. Nowadays three different schemes exist to enhance the shear strength of rectangular or prismatic elements (Fig. 9).

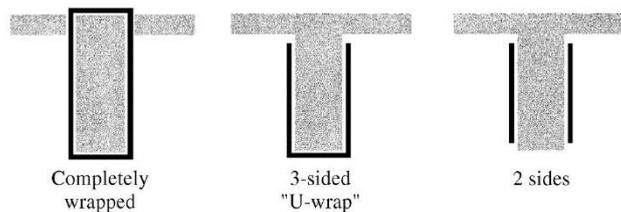


Fig. 9 Shear wrapping schemes

Following the (ACI-Committee-440 2008) prescriptions “completely wrapping the FRP system around the section on all four sides is the most efficient wrapping scheme and is most commonly used in column applications where access to all four sides of the column is usually available. In beam applications where an integral slab makes it impractical to completely wrap the member, the shear strength can be improved by wrapping the FRP system around three sides of the member (U-wrap) or bonding to two opposite sides of the member. Although all three techniques have been shown to improve the shear strength of a member, completely wrapping the section is the most efficient, followed by the three-sided U-wrap. Bonding to two sides of a beam is the least efficient scheme. In all wrapping schemes, the FRP system can be installed continuously or placed as discrete strips”.

The effectiveness of shear strengthening intervention is influenced by FRP sheets geometry (thickness, width, space) and by the inclination of composite strips (i.e. fiber orientation) with respect to RC element longitudinal axis, expressed by the coefficient α . “The shear contribution of the FRP shear reinforcement is then given by”:

$$V_f = \frac{A_{fv} f_{fe} (\sin \alpha + \cos \alpha) d_{fv}}{s_f} \quad (16)$$

Where:

$$A_{fv} = 2nt_f w_f \quad (17)$$

$$f_{fe} = \varepsilon_{fe} E_f \quad (18)$$

In eq. (17) and in eq. (18) s_f is the space between two adjacent strips, w_f is the width of the FRP strips, t_f is the strips thickness, n represents the number of FRP plies and E_f and ε_{fe} are respectively the FRP Young modulus and the FRP effective strain.

3.4.3- Confinement

An appropriate confinement of RC columns by means of FRP jackets can determine an improvement of the performances, in terms of concrete compressive strength and ductility. The increase of the concrete strength is related to the lateral confinement pressure: indeed, “FRP jackets provide passive confinement to the member, remaining unstressed until dilation and cracking of the wrapped compression member occur”. Therefore, once that the RC element reaches the plasticization, identified with the cracking of the RC element, the lateral pressure increases and with it also the effectiveness of the confinement on the concrete strength.

The effectiveness of the confinement is related with the cross-section shape: in fact, it will be greater for circular sections and lower for rectangular or square sections, where the right angles have to be smoothed to enhance the effects of confinement. While, for circular cross-sections the (ACI-Committee-440 2008) established that “FRP system provides a circumferentially uniform confining pressure to the radial expansion of the compression member”, increasing the effectiveness.

Confining a RC element with FRP jackets orienting the fibers transverse to the longitudinal axis causes an enhance of compressive strength, acting like tie reinforcing steel. The behaviour of the strengthened section depends on the level of confinement, as shown in (Fig. 10): the bigger the confinement action, the bigger the strain capacity.

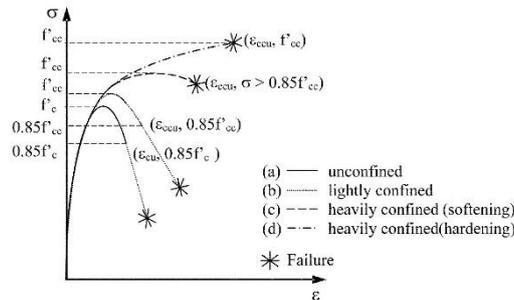


Fig. 10 Stress-strain behavior of unconfined and confined RC columns (Rocca et al.2006)

The compressive strength for the confined element is:

$$N_{Rec,d} = \frac{1}{\gamma_{Rd}} \cdot A_c \cdot f_{ccd} + A_s \cdot f_{yd} \quad (19)$$

Where γ_{Rd} is a partial safety factor equal to 1.10 A_c is the area of the transversal concrete cross-section and f_{ccd} is the tensile design strength of confined concrete. A_s is the area of the steel reinforcement and f_{yd} its design tensile strength.

The peak concrete strength is calculated as:

$$\frac{f_{ccd}}{f_{cd}} = 1 + 2.6 \cdot \left(\frac{f_{l,eff}}{f_{cd}} \right)^{2/3} \quad (20)$$

Where $f_{l,eff}$ represents the effective lateral pressure.

Just a rate of the confinement pressure, called effective confinement pressure, contributes to the strength of confined element, and it depends on the cross-section shape and on the intervention mode:

$$f_{l,eff} = k_{eff} \cdot f_l \quad (21)$$

The confinement pressure is calculated as:

$$f_l = \frac{1}{2} \rho_f E_f \varepsilon_{fd} \quad (22)$$

Where ρ_f represents the geometric strengthening percentage and depends by the cross-section shape and on the intervention mode, and for rectangular section is equal to:

$$\rho_f = \frac{2 \cdot t_f \cdot (b+h) \cdot b_f}{b \cdot h \cdot p_f} \quad (23)$$

Where p_f represents the distance between two adjacent FRP sheets, b_f is the width of FRP sheets and t_f is the thickness of FRP sheets.



Fig. 11 Discontinuous confinement of a circular section

The effectiveness coefficient k_{eff} is the product of three coefficients related to the:

- Cross-section shape (k_h)
- Development of confinement along the longitudinal axis of the strengthened element (k_v)
- Orientation of the fibers (k_α)

In case of a complete jacketing of a rectangular RC element the three parameters are calculated as:

$$k_v = 1 \quad (24)$$

$$k_\alpha = \frac{1}{(1 + \operatorname{tg}(\alpha)^2)} \quad (25)$$

$$k_h = 1 - \frac{b^2 + h^2}{3A_g} \quad (26)$$

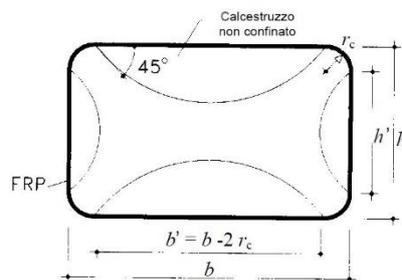


Fig. 12 Confinement of a rectangular section

As shown in (Fig. 12) the effective area of confined concrete does not correspond to the entire cross-section area, due to the amplitude of the rounding angle: the bigger the angle the bigger the confined area. Finally, the effect of the confinement is proved just for those section for which the ratio b/h is greater than 2 and with the maximum dimension smaller than 900mm.

All the reinforcement systems described above can be usefully used to intervene on reinforced concrete structures within a seismic zone, by removing brittle mechanisms of collapse (i.e. shear failure) and enhancing the global strain capacity of the structure, by replacing the plastic hinges in according to the hierarchy of resistance or increasing the rotational capacity of the plastic hinges without changing their position.

3.5- Optimization

In the engineering field the design or analysis procedure is usually iterative and time consuming. In general, the solution must be found in a reasonable interval of time. According with (Vanderplaats 2001) “In engineering the purpose is to produce the “best quality of life possible with the resources available””. To achieve this purpose, thanks to the spread of computers, the numerical optimization can be the right and only one way to obtain it. First, when dealing with optimization it is fundamental to distinguish analysis from design. Through the first process the response of a system to its environment is determined, while through the second one the system is properly defined. Optimization process, since the analysis is a design subprocess, is particularly used to find the best design.

Nevertheless, numerical optimization is often used also for analysis as well: in particular it is widely used in the civil engineering field for performing nonlinear analysis, where the analysis time to achieve a solution is often high, thanks to the great number of variables. Fortunately, much of the design variables are quantifiable, so computers can be used to analyze alternatives rapidly.

Usually the design path is reduced to a succession of analytical steps that help the designer to establish the best solution. This procedure can be effective when the complexity is limited and otherwise is not possible to find an analytical solution. Nowadays, the designer are helped in the design process by computer codes that are, according with (Vanderplaats 2001) “capable of analyzing a design which the engineer considers reasonable. With the availability of computer codes to analyze the proposed design, the next logical step is to automate the design process. In its most basic form, design automation may consist of a series of loops in the computer code which cycle through many combinations of design variables. The combination which provides the best design satisfying the constraints is then termed optimum. This approach has been used with some success and may be quite adequate if the analysis program uses a small amount of computer time. However, the cost of this techniques increases dramatically as the number of design variables to be changed increases and as the computer time for a single analysis increases.”

If the number of variables is high and the time to test each variable is great, the total analysis time can be overstated, so a more rational approach must be considered. This leads to numerical optimization techniques, that offer a logical approach to automation. Many algorithms have been developed in recent years, “such as linear, quadratic, dynamic and geometric programming algorithms. A more general category of algorithms referred to as nonlinear programming has evolved for the solution of general optimization problems”.

Among these there are the Evolutionary Algorithms (EA), that are biology-inspired methods used for global optimization. They are inspired by biological evolution, and so gives to them this name. Among the EA there are the Genetic Algorithms (GA), that seek the solution in the form of string of numbers, which are binary in the classic formulation (CGA).

The GA works, in each iteration, on a set of solutions which are the possible global solutions to the optimization problem. The fitness of each solution is evaluated through the fitness function, that is defined in a space that includes all the possible solutions. Clearly, GA can be used in both minimization or maximization problems, to whom the fitness will be the lowest or the highest respectively.

Preliminarily, in the *encoding* portion of the problem, the form of solutions is defined: usually they are represented by bit strings in the Canonical Genetic Algorithms (CGA). After that the population is defined, through the initialization of the solutions. This step is up to the algorithm developer: thanks to the knowledge of the problem, he can wisely choose them.

Each algorithm is generally composed by six portions, that will be described below, in according with (Ferrari 2018) and which can be summarized as follows:

| A simple generic Algorithm |
|---|
| 1: Initialize population |
| 2: repeat |
| 3: repeat |
| 4: Crossover |
| 5: Mutation |
| 6: Phenotype mapping |
| 7: Fitness computation |
| 8: until population complete |
| 9: Selection |
| 10: until Termination conditions met |

Table 2 Generic GA workflow

“The sexual reproduction goes through a genetic crossover between two parents chosen from the population, a random mutation in the genes and the fitness calculation of new offspring. This cycle is iterated until the chosen number of candidate solutions in the population is reached”.

3.5.1- Crossover

In nature the reproduction can be sexual or asexual: in the first case two organisms mix their genetic material to create a new offspring, while in the second case the reproduction occurs in the form of genetic cloning. Since through the GA an evolution of solution is sought, the GA use a methodology that corresponds to sexual

reproduction. In fact, according with (Ferrari 2018) “two or more candidate solutions from the parental population mix their genetic information to breed new offspring for each reproduction cycle”.

In general, the parents are chosen from the parental population randomly, picking them through uniform probability or through a distribution proportional to each candidate solution’s fitness. After that, the candidate solutions’ genes must be mixed: this operation is called Crossover precisely because the offspring is generated segmenting and crossing over the two parents’ genetic information.

Many crossover methods exist, such as n-point crossover, one-point crossover and uniform crossover. “The first one chooses n bit positions that split the two solutions in n + 1 bit segments; then it swaps every other segment of one parent with the corresponding segment of the other parent. This method breeds two new transitory offspring (still without mutation) that inherit complementary parts of the two parents”.

The second one chooses randomly a position c within the string length and swaps all bits with index < c from the parents. “For example, if 01010101 and 11110000 are the two parents and is picked the 7th position, then the o_springs will be 010101-00 and 111100-01”. One-point crossover is affected by epistasis or linkage: in fact, farther elements of the parental solutions have a higher probability of being separated than closer ones.

The linkage/epistasis issue is overcome through the third methods, called Uniform Crossover: the linkage is deleted by crossing over each index’s elements independently of one another.

3.5.2- Mutation

Within the Evolution Theory, mutations are a modification of some information in the genome of the organism. It is a fundamental process also for GA as it alters the genes of a solution: the more the mutation affects the solutions, the higher the so-called mutation rate. Without mutation there is the possibility that all the genes of the population are the same (premature convergence); on the other hand, a high mutation rate leads to a chaotic search. In order to reach the right proportion of exploration over exploitation, the mutation operator of the algorithm must be scalable: in other words, it's important that the mutation rate can be adjusted. “Another requirement for mutation operator is reachability; it requires that every point in the solution space is reachable from any other point through the algorithm”. The nature of the problem determines the implementation of the mutation operator: “If the points of the solution space are bit strings, the mutation consists in single bit flip; usually a Bernoulli distribution with parameter $p = p_m$ or $p = 1/n$ (with $n =$ population size) is used for each bit in the solution”.

3.5.3- Genotype – phenotype mapping

The *genotype* and the *phenotype* help to assess the offspring’s fitness for the optimization problem. All the genes that define and form the solution in the reproduction cycle constitute the genotype. The corresponding individual is called phenotype. After the crossover and the mutation, an eventual mapping of genotype and phenotype allows the successive evaluation step.

3.5.4- Fitness

When assessing the quality of a solution, it is better to distinguish between *evaluation function* and *fitness function*. “The former evaluates the quality of the solution's phenotype with respect to a particular set of parameters. The fitness function, instead, returns an indication on the reproductive quality of the solution, i.e., its fitness. To do so, it usually considers the evaluation of the solution with respect to the average evaluation of the population”. The designer of the GA must define properly the design of the fitness function, with particular regard to the selection step. To do so there are two main approaches: through the first one the population is evaluated and the first n solutions evaluation-wise are selected: so, the sorting of population is required. Through the second approach the solutions whose fitness is greater than a predefined index is selected. “Assessing a quality value to the solutions and deciding which of them survives for the successive generation are strictly bound parts of the algorithm's design”.

3.5.5- Selection

When a generational cycle ends, “the selection operator decides which solutions survive the next generation and which do not”. It seems obvious, for the GA philosophy, that the fittest solutions should be favoured in order to allow the evolution of the population, as mentioned by Darwin on his survival of the fittest principle. To pursue this aim there are many methods, among these, three are the most relevant and used:

- **Elitist Selection:** through this selection method the fittest offspring is selected for the next generation;
- **Comma selection:** this method is a sub-class of the Elitist Selection. The next generation's parental population are formed by all the fittest individuals among the generated offspring, but none of the parents will survive in the next cycle;
- **Fitness Proportionate Selection (FPS):** in this case the individuals are picked in proportion to their fitness;
- **Tournament Selection:** this method consists of two steps: in the first one a subset of the population is randomly selected and on the second step the next generation's individuals are picked using for instance the FPS method.

With the last two methods every individual has the same chance to be selected, but at the same time the fittest individual might not be selected.

3.5.6- Termination

The termination conditions concern the main cycle terminates. They are usually linked with time or number of cycles bounds, to avoid infinite search. Restart strategies should be expected to look for other solutions if the search gets stale. If restarting from different areas in the solution space leads to the same local optimum, it is a justified assumption that the local optimum is a large attractor, or indeed the global optimum.

3.6- FRP and Optimization

While many studies on the behavior of FRP-confined RC elements have been carried out, especially for what concerns the behavior of circular concrete columns strengthened with FRP, the corresponding studies for investigating the optimized disposition strategies of FRP on real scale buildings are almost unknown.

Zou, Teng et al. (2007) established a methodology that aimed at defining the optimum usage of FRP as strengthening strategy of existing RC buildings. Their work was based on the assessment of one variable, the FRP thickness to which they refer as design variable.

Their study aims to minimize the volume of FRP employed and consequently to minimize its cost. The building performance was ranked in according with the pushover drift, expressed as a function of the design variable through the principle of virtual work and the Taylor series approximation. In their study they consider the installation of FRP sheets just where the plastic hinges are expected to develop, i.e. at the elements ends.

The two equations that determines the design regards respectively the cost and the lateral drift:

$$\cos t = \sum_{i=1}^{N_c} w_i t_i \quad (27)$$

$$u_j(t_i) = u_j \Big|_{t_i=t_i^0} + \sum_{i=1}^{N_c} \frac{\partial u_j}{\partial t_i} \Big|_{t_i=t_i^0} (t_i - t_i^0) + \frac{1}{2} \sum_{i=1}^{N_c} \frac{\partial^2 u_j}{\partial t_i^2} \Big|_{t_i=t_i^0} (t_i - t_i^0)^2 \quad (28)$$

Both equations depend on the design variable. In eq. (27) w_i represents the cost coefficient for the FRP composite, defined as $w_i = 4L_{ci}(B_i + D_i)\rho$ and ρ is the cost per unit volume of FRP. Through eq. (28) the lateral drift is obtained using the principle of virtual work and the Taylor series approximation. Eq. (28) defines the lateral drift at a given story, so, for the optimization algorithm is used the function $g_j(t_i)$ that expresses the difference in terms of lateral drift between two different floors (eq. (29))

$$g_j(t_i) = \frac{1}{h} \left[\Delta u_j \Big|_{t_i=t_i^0} + \sum_{i=1}^{N_c} \beta_{1i} (t_i - t_i^0) + \frac{1}{2} \sum_{i=1}^{N_c} \beta_{2i} (t_i - t_i^0)^2 \right] \leq d_j^U, \quad (j = 1, 2, 3, \dots, N_j) \quad (29)$$

Once that the problem is explicitly defined a suitable method to solve it must be adopted: in the Zou et al. study, the Optimality Criteria (OC) is chosen thanks to its numerical efficiency for the design of large-scale structures. When using the OC the model must be re-analyzed at each cycle, then the continuous optimization process is reapplied until convergence in the cost optimization is achieved.

Through the study of Zou, Teng et al. (2007), an optimization technique for the retrofit design of RC is presented, but it considers basically an optimization in terms of costs, while the search of an optimized disposition of FRP on the columns that must be retrofitted seems neglected.

3.7- OpenSees

OpenSees (Open System for Earthquake Engineering Simulation) is an open source software worldwide used for structural analysis in the academic field. This was the main reason which led to choosing it for performing the analysis. It was developed by Franck McKenna at the PEER. Since it is not commercialized, the application lacks an interface and the user has to create scripts to define the objects of the assessment (Fig. 13).

```
167 eleLoad -ele 363 -type -beamUniform -$Qd1Beam 0.;
168 eleLoad -ele 373 -type -beamUniform -$Qd1Gird 0.;
169 eleLoad -ele 383 -type -beamUniform -$Qd1Beam 0.;
170 eleLoad -ele 393 -type -beamUniform -$Qd1Gird 0.;
171 eleLoad -ele 403 -type -beamUniform -$Qd1Beam 0.;
172 }
173 set TestType EnergyIncr;
174 set Tol 1.e-03;
175 set maxNumIter 100;
176 test $TestType $Tol $maxNumIter;
177 constraints Transformation;
178 numberer Plain;
179 system BandGeneral;
180 algorithm KrylovNewton;
181 set NstepGravity 10;
182 set DGravity [expr 1./$NstepGravity];
183 integrator LoadControl $DGravity;
184 analysis Static;
185 set Gravity [analyze $NstepGravity];
186 if {$Gravity != 0} {
187     puts "Gravity loads not OK"
188 }
189 loadConst -time 0.0
190
191 puts "Analysis: NLTHA"
192 # Set up the acceleration records for Norcia fault normal and fault parallel
193 set ScaleFactor 9.81;
194 set NorciaEW "Path -filePath Norcia_EW.txt -dt 0.005 -factor $ScaleFactor";
195 set NorciaNS "Path -filePath Norcia_NS.txt -dt 0.005 -factor $ScaleFactor";
196 # Define the excitation using the Norcia ground motion records
```

Fig. 13 OpenSees Script example

The software consists of an interpreter for TCL or Python programming language and it incorporate the seismic engineering and the finite element method. The great variety of elements and materials provided by the software allow the user to test an infinite number of configurations. Finally, its versatility makes it particularly suitable for performing non-linear analysis: the inelasticity can be checked at the material, element and analysis type stage.

4. METHODOLOGY

The first usage of OpenSees for a beginner is not simple. The user has to face with a multitude of command with their specific syntax and with a programming language (i.e. Tcl) that is not immediate, in fact it differs quit a lot from the syntax used for programming in Matlab environment or in Python. Moreover, the lack of an interface, which is instead fundamental in the commercial software, such as SAP2000, does not allow a real time check of the progresses, causing errors and spending many times to build the model.

To avoid these issues a simple elastic reinforced concrete one story frame was modelled both in SAP2000 and in OpenSees. The aim of this thesis is to analyze a five story RC frame: a model of such dimensions is not easy to handle, so all the steps were repeated for five different frames, from one to five story. It is a time-consuming procedure, but necessary to overcome the faults and to know how OpenSees works.

4.1- SAP2000 elastic model building

In SAP2000 the user must define a cross section which is assigned to the elastic element. If the section is composed by multiple materials, e.g. steel rebars for reinforced concrete members, the reinforcement will not

be considered for the stiffness calculation. All the frames were thought to behave as shear type frame and built following the approach of lumped mass. Shear type frame consists of infinite rigid diaphragm at each story, that behave as a rigid body, thus differential shifts are neglected. The point located in the middle of each story is named centre of mass: in fact, all the story mass is lumped in this point, where all the external forces will be applied.

4.2- Opensees elastic model building

In OpenSees the elastic elements (i.e. *elasticBeamColumn*) for three-dimensional model are defined by some cross-sectional properties, such as: area, modulus of Elasticity, shear modulus, moment of inertia of cross-section and the two second moment of inertia I_z and I_y . The rigid diaphragm is assigned through the command *rigidDiaphragm*, that define a master node (i.e. the centre of mass) and a list of slave nodes that move as if in a rigid plane with the master node. Moreover, the mass of each story is simultaneously applied to the master node.

4.3- Eigenvalue analysis

Several eigenvalue analyses were performed for both models in order to compare to each other. In this analysis type the dynamic response of the structure subject to a free vibration is obtained. The equation of motion of a Multi Degree Of Freedom (i.e. MDOF) not subjected to damping is:

$$[M] \cdot \left\{ \ddot{u}(x,t) \right\} + [K] \cdot \{u(x,t)\} = \{0\} \quad (30)$$

This linear system in matrix form provides coupled equation of motion: to overcome this issue a coordinates transformation is applied:

$$\{u(x,t)\} = [\phi(x)] \cdot \{q(t)\} \quad (31)$$

The coordinates transformation led to another matrix system, depending on the mass matrix and on the stiffness matrix:

$$[\overline{M}] \cdot \left\{ \ddot{q}(t) \right\} + [\overline{K}] \cdot \{q(t)\} = \{0\} \quad (32)$$

This system is uncoupled only when the $[\overline{M}]$ and $[\overline{K}]$ are diagonal

$$[M_D] = \{\phi\}^T [M] \{\phi\} \quad \text{and} \quad [K_D] = \{\phi\}^T [K] \{\phi\}$$

Where the vector $\{\phi\}$ represent the eigenvector. Replacing the dynamic response in terms of displacement and acceleration in the eq.(30), the new equation is obtained

$$(K_D - \omega^2 M_D)[\{\phi\}] = \{0\} \quad (33)$$

The only meaningful solution is obtained from the following equation, that summarize the eigenvalue problem.

$$\det[[K_D] - \omega^2 [M_D]] = 0 \quad (34)$$

The eq.(34) is called characteristic equation in which $[M_D]$ and $[K_D]$ are known and ω^2 represents the eigenvalue, that are in the same number of the degrees of freedom of the structure. According with (Humar 2012) “a vibrating system with N DOFs has N natural vibration frequencies ω_n ($n= 1, 2, \dots, N$), arranged in sequence from smallest to largest, corresponding natural periods T_n and natural modes ϕ_n . The term natural is used to qualify each of these vibration properties to emphasize the fact that these are natural properties of the structure in free vibration, and they depend only on its mass and stiffness properties. The first mode is also known as the fundamental mode”.

4.4- Elastic response spectrum

In Seismic Engineering there is a useful tool, called Elastic Response Spectrum, that allows to represent the maximum expected seismic action on a structure. Each spectrum provides a relation between the maximum seismic action on the building, in terms of maximum displacement, velocity or acceleration, and its fundamental period, obtained for a given ground motion acceleration history. This methodology is general, so it can be applied to any structure once that its mass and its stiffness are known.

The response spectrum (Fig. 14) is available in form of graph, in which in the vertical axis there is the parameter chosen to describe the maximum of the seismic action, while in the horizontal axis is reported the natural period, generally expressed in seconds.

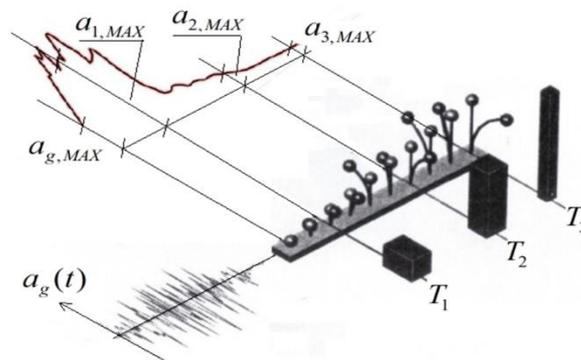


Fig. 14 Response Spectrum construction

Each value on the x-axis represents the fundamental period of a well-established building and the value that intersects in the vertical axis represents its maximum response to a given strong motion. Moreover, the higher the value of the period the lower the stiffness of the building is.

The Response Spectrum can be obtained by numerical integration of the motion equation for each oscillator inside the elastic field (eq.(35))

$$M\ddot{u} + c\dot{u} + ku = -Ma_g \quad (35)$$

In eq.(35) M , c and k represent respectively the mass, the viscous damping and the stiffness of the considered oscillator; u and its first and second derivatives represent the displacement, the velocity and the acceleration at a generic time instant and a_g is the seismic acceleration applied at the structure.

Observing eq.(35) it is possible to deduce a relation displacement-acceleration for an undamped system

$$U = \left(\frac{k}{m} \right) \cdot u = (\omega^2) \cdot u \quad (36)$$

Where U is the absolute acceleration and ω the angular frequency. The eq.(36) can be written as:

$$S_a(\omega) = \omega^2 S_d(\omega) \quad (37)$$

In eq.(37) on the first member is defined the generic spectral ordinates of the acceleration response, but since this formulation is available for undamped system, it is identified as an ideal case and for that it is usually called pseudo-acceleration spectrum. Using the Duhamel integral, it is possible to obtain the corresponding pseudo-velocity response spectrum, which leads to a relation similar to eq.(37), but with the generic spectral ordinates of the velocity response at the first member (eq.(38))

$$S_v(\omega) \cong \omega S_d(\omega) \quad (38)$$

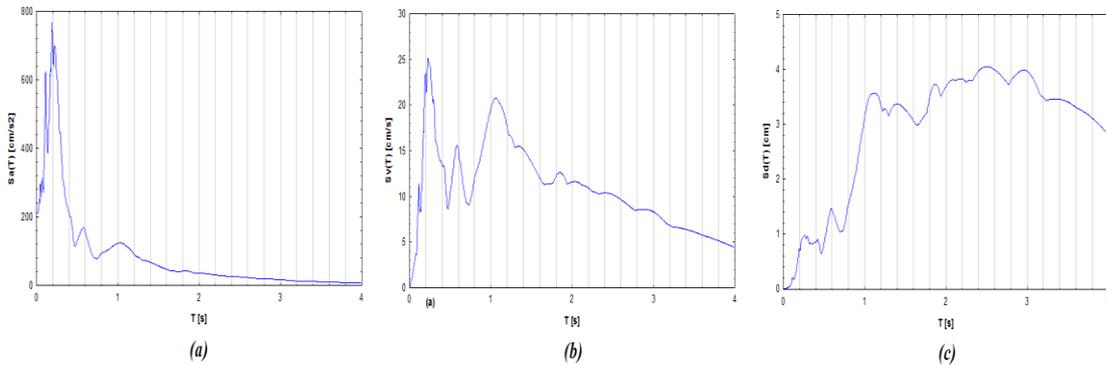


Fig. 15 Acceleration, velocity and displacement response spectra for Kozani-prefecture record

4.5- Uniform hazard spectrum UHS

Different types of response spectra are available in literature, among these there is the Uniform Hazard Spectrum in which every spectral acceleration has the same return period. Each spectrum is provided for a given exceedance probability: a UH spectrum built for an exceedance probability of 2% in ten years means that there is the 2% of probability that in the next ten years an earthquake of magnitude greater than that corresponding to UHS will not occurs. Each UHS is correlated with a specific seismic event through the hazard deaggregation.

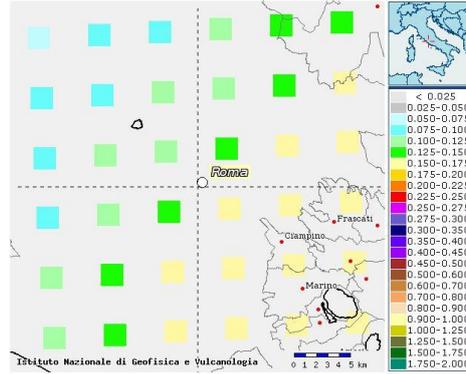


Fig. 16 Hazard Deaggregation for the city of Rome

The aim of the UHS is of designing a building with the same risk level, and according with (Cimellaro and Marasco 2016) it “means considering ground motions parameters with equal exceedance probability independent from the natural period of the building and from the given seismic event”.

4.6- Design Response Spectrum

The Design Response spectrum are different for each nation, according to the corresponding national code. In Italy is used the DS provided by the NTC2018, which is inspired by the European Code (EC8).

Each DS is defined starting from the exceedance probability (P_{VR}) and from a period (V_R). The second parameter is related with the “life” of the building that is going to be designed: it expressed its durability and importance. Thus, V_R represents the time period in which the structure must maintain its serviceability and functionality. The NTC2018 provides an equation for calculating V_R :

$$V_R = V_N \cdot C_u \quad (39)$$

Where V_N represents the design life of the structure and C_u is related with its importance class. The most important parameter for the definition of the DS is the return period T_R (eq.(40)). Through this variable, to all the DS building procedure, a probabilistic approach is considered.

$$TR = \frac{-V_R}{\ln(1-P_{VR})} \quad (40)$$

Moreover, the DS changes according to the considered Limit State: in fact, at each LS, grouped in two classes (i.e. Serviceability and Ultimate), is assigned an exceedance probability. The DS shape instead, changes according to three parameters:

- Maximum ground acceleration a_g
- Maximum amplitude factor for horizontal acceleration spectrum F_0
- Initial period value of constant velocity range T^*c

Each of the mentioned parameter is provided by INGV for a given exceedance probability: in particular they are calculated for nine different values of return period. These three coefficients are obtained for a flat and rigid ground, so exist two other parameters that take into account the amplification phenomena due to stratigraphy effects and topographic effects; they are:

- Shear wave velocity measured at 30 meters deep (VS30)
- Topography amplification coefficient

The VS30 changes proportionally according to the stiffness of the ground: the greater the stiffness the greater the value of VS30. The second coefficient depends on the conformation of the soil and has its maximum for a building designed on the top of a slope.

Moreover, to build the DS is necessary to determine the behavior factor q : through this parameter the DS is scaled to take into account the nonlinearities of the frame that confer to it a dissipative behavior. Thus, the application of q leads to a reduction of the seismic action. Below the procedure to determine q is shown:

$$q_{lim} = q_0 \cdot K_R \quad (41)$$

Where q_{lim} represents the maximum value for q . The coefficient q_0 depends on the structural type and on the ductility class, while K_R depends on the regularity in height of the structure.

Once that all of these coefficients are calculated, the DS can be obtained.

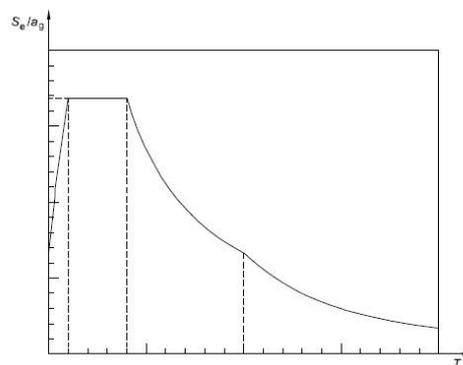


Fig. 17 Design Spectrum typical shape

4.7- SAP2000 non-linear model building

Once that the elastic properties of the two models correspond it is possible to introduce the non-linearities, concerning the materials and the elements.

If in the elastic model the rebars were not taken into account, for non-linear analysis the user must define also the reinforcement, in terms of number of bars and steel type. This is achieved modelling the section through the section designer, a SAP2000 tool that allows to design the RC elements cross section. As for concrete, the user can define a new type or choose among several steel types, suggested by the international codes. So, the material non-linearities are considered.

When considering the lumped plasticity approach the plastic hinges are employed: their usage means that all the inelastic effects are in a well-established position of the reinforced concrete columns and beams. For a shear type frame they are located at the elements ends. SAP2000 allows the user to set different plastic hinge types, based on the element on which they arise and on their stress-strain relationship. For the columns the P-M2-M3 type was chosen while for the beams the M2-M3 type. The fundamental parameter to set for the plastic hinge is its length: it describes the length in which the user chose to lump the plastic behaviour at both ends: 0.05 and 0.95 are usually used. The two values mean that the hinge length will be created at the 5% and at the 95% of the element length from the connection point.

4.8- Opensees non-linear model building

The first step for building an inelastic model is the selection of the materials. OpenSees supplies a wide assortment of materials, that are useful to define the cross-section of the elements. Each material is represented in the software by its stress-strain relationship, that describe its behaviour for typical situation, so that the material that best fits with the expected structural behaviour can be selected. The greater part of the material formulation comes from experimental tests carried out on specimens subjected to a specific load path or realized by new technologies.

The second step concerns the element formulation. As mentioned above there are two approaches that describe the definition of the plasticity. For the case of lumped plasticity the non-linearity was accounted in a specific element area, commonly known as plastic-hinge. The main issue when using this formulation is the determination of the hinge length, but nevertheless it is less time consuming.

For the distributed plasticity approach the inelasticity is spread along the element length and not restricted to a specific area at the ends. When dealing with large inelastic model, this approach is more time consuming and can lead to convergence issues.

According with (Terzic 2011) OpenSees provides a set of elements suitable for both approaches. There are two main classes of elements: displacement-based element (DB) and force-based element (FB). The DB elements can be used just for the distributed plasticity approach and for their formulation the elements are imposed with a displacement fields: thus, the equilibrium equations are solved depending on the stiffness.

For the FB elements instead, a force and moment fields are imposed: the equilibrium for the governing equations is reached by means of flexibility. This element formulation was realized for distributed plasticity model, but there is an adaptation for lumped plasticity approach that includes the definition of plastic-hinge length at the ends of the elements. According to (Spacone, Filippou et al. 1996) the (Scott and Fenves 2006) “force-based beam-column elements are formulated in a basic system without rigid-body displacement. The element deformations, in the vector \mathbf{v} , are assumed small compared to the element length. From the principle of virtual forces, the compatibility relationship between the section and element deformations is”:

$$\mathbf{v} = \int_0^L \mathbf{b}^T \mathbf{e} dx \quad (42)$$

Where the matrix \mathbf{b} “contains interpolation functions relating section forces to basic forces from equilibrium of the basic system. The compatibility relationship is evaluated by numerical quadrature:

$$\mathbf{v} = \sum_{i=1}^{N_p} (\mathbf{b}^T \mathbf{e}|_{x=\xi_i}) \omega_i \quad (43)$$

where ζ and ω are equal to locations and associated weights, respectively, of the N_p integration points over the element length $[0, L]$. Gauss–Lobatto quadrature is used in force-based elements because it places integration points at the element ends, where the bending moments are largest in the absence of member loads. At least three Gauss–Lobatto integration points are required to represent exactly a linear curvature distribution along the element. To represent accurately the nonlinear material response of a force-based beam–column element, four to six Gauss–Lobatto integration points are typically used”.

4.9- Non-linear elements with plastic hinge

OpenSees provides an inelastic element type called *forceBeamColumn* that allow the user to set also the plastic hinges properties, after establishing its length. According with (Scott and Fenves 2006) “The plastic hinge length for FBE is related to the integration method.”[...] “This causes a loss of objectivity for section softening behavior: the deformation is localized in a single integration point, so a unique solution does not exist and the response depends on the integration method”. (Spacone, Filippou et al. 1996) and (Paulay and Priestley 1992) provided two different equations for determining the plastic hinge length, where this quantity is treated as a free parameter.

$$l_p = \frac{G_f^c}{0.6 f'_c (\varepsilon_{20} - \varepsilon_c + 0.8 f'_c / E_c)} \quad (44)$$

$$l_p = 0.08L + 0.022 f_y d_b \quad (kN, mm) \quad (45)$$

All the plastic hinge integration methods are based on the idea that the nonlinear behaviour is limited to the regions of length L_{pI} and L_{pJ} at the element ends. The compatibility equation (eq.(42)) is split into three integrals to represent plastic hinge for FBE, one for the interior region and the other two for the element ends:

$$v = \int_0^{l_{pI}} b^T e dx + \int_{l_{pI}}^{L-l_{pJ}} b^T e dx + \int_{L-l_{pJ}}^L b^T e dx \quad (46)$$

“The section deformations are integrated numerically over the plastic hinge regions, whereas the contribution of the element interior is assumed to be linear elastic and evaluated by the flexibility of the interior region

$$v = \sum_{i=1}^{N_p} (b^T e|_{x=\zeta_i}) \omega_i + f_{\text{int}}^e q \quad (47)$$

where ζ and ω are equal to locations and associated weights, respectively, of the N_p integration points in the plastic hinge regions”.

Within this thesis work the modified Two-Point Gauss-Radau Integration method is used (Fig. 18): the two-point of Gauss-Radau integration rule are applied at $4L_{pI}$ and $4L_{pJ}$ from the element ends, giving the followings integration point locations and weights:

$$\zeta = \{0, 8l_{pI}/3, L - 8l_{pJ}/3, L\} \quad (48)$$

$$\omega = \{l_{pI}, 3l_{pI}, 3l_{pJ}, l_{pJ}\} \quad (49)$$

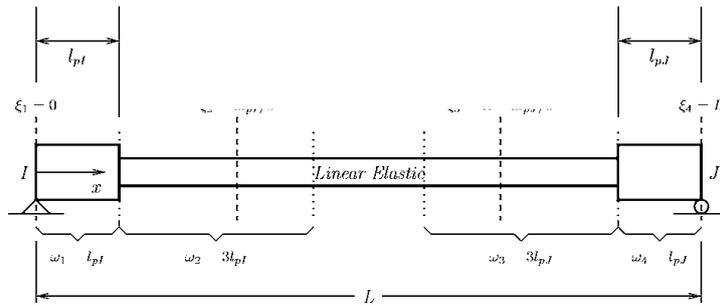


Fig. 18 Modified Gauss-Radau plastic hinge integration methods

Through this approach the nonlinear behaviour is confined at the element ends assuming for the interior element section a linear elastic behaviour. Therefore, according with (Scott and Fenves 2006) “plasticity is confined to a single integration point at each end of the element”.

All these studies come together in OpenSees through the definition of the *Hinge-Radau* integration method for plastic hinge, in which, in according with (Scott 2011) “the plastic hinge length at end I (J) is equal to l_{pI} (l_{pJ}) and the associated force deformation response is defined by the section with tag secTagI (secTagJ). The force

deformation response of the element interior is defined by the section with tag secTagE. Typically, the interior section is linear-elastic, but this is not necessary”.

4.10- Non-linear analysis

Once that the inelastic models are built they can be submitted to the analysis. Two different analysis type were selected: non-linear static analysis and non-linear dynamic analysis.

4.11- Algorithm and integrators

In both Opensees and SAP2000 the nonlinear equations are solved through an iterative solution procedure using the Newton-Raphson method, that “is the most widely and the most robust method for solving nonlinear algebraic equations”, following the (McKenna 2011) prescriptions. The time integration, for the time history analysis, is pursued in both software using the Newmark method, but in OpenSees, since the nonlinear model is complex, to force the convergence other integrators are used, such as Broyden or KrylovNewton, building an adaptive procedure.

4.12- Non-linear static analysis

The non-linear static analysis is commonly called pushover analysis, because the structure is literally pushed by a system of lateral forces. This analysis type is widely used in civil engineer field to assess the seismic vulnerability of a structure and it is based on its performance. The purpose of this assessment is to obtain a force-displacement relationship that represent the capacity of the structure, monitoring the displacement of the control point, that usually coincides with the centre of mass of the last story of the building. During the pushover analysis the gravity loads are constant while horizontal loads, representing inertial force, are increasing monotonically until a horizontal target displacement is achieved: in fact, it represents the maximum displacement expected during the design earthquake. The distribution of lateral forces represents the seismic action and are provided by the elastic design spectrum: in according with some parameters, it provides the spectral acceleration, that is the maximum acceleration expected on the structure.

The Finite Element model must incorporate the non-linear stress-strain relationships of those elements that are expected to pass the elastic limit.

In according with UNI-EN (1998) at least to two lateral force distributions must be applied to the structure, and they are defined as:

- 1- Uniform distribution based on the lateral forces proportional to the mass neglecting the height of the structure;
- 2- Distribution proportional to the first mode, provided by the linear dynamic analysis.

The (NTC 2018), suggests using two force distributions; all the possible distributions are divided in two groups and below both are described. They come respectively from the first and from the second group:

- Proportional to the following static forces, if the first natural period is preponderant:

$$F_i = F_h \cdot z_i \cdot \frac{W_i}{\sum_j z_j W_j} \quad (50)$$

Where

$$F_h = S_d(T_1) W \lambda / g \quad (51)$$

W_i and W_j are the weight corresponding to the mass I and j , z_i and z_j are the height from the ground floor of the masses I and j , $S_d(T_1)$ is the ordinate of the response spectrum, W is the total structure weight and g is the gravity acceleration. T_1 is defined, for civil constructions up to 40 meters high, as:

$$T_1 = \sqrt{2d} \quad (52)$$

Where d is the lateral displacement of the higher point of the building, due to the following load combination applied in horizontal direction:

$$G_1 + G_2 + \sum_j \psi_{2j} Q_{kj} \quad (53)$$

- Force distribution deduced from a uniform distribution of accelerations along the structure height.

The results of the pushover analysis are expressed in terms of capacity curve.

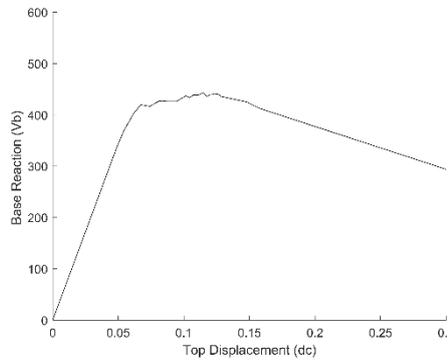


Fig. 19 Capacity curve

This curve records step by step the base shear (V_b) and the displacement of a particular point located on the top of the structure (d_c). On this plot the ultimate limit state can be recognized.

This analysis is effective for regular buildings, since it is based on the fundamental period that means that through this assessment the structure is reduced to an equivalent single degree of freedom. Nevertheless, due to the static nature of the method the dynamic effects on the structure cannot be captured.

4.13- Non-linear dynamic analysis

The non-linear dynamic analysis, also called Time History analysis, is the most effective analytical method to represent the dynamic response of a structure. It needs of a non-linear finite element model, in which the material non-linearities are well-accounted. Performing a time history analysis is a time-consuming procedure and it is also complicated. Nevertheless, is requested when the non-linear behaviour of a structure must be investigated. The model is subjected to an earthquake loading represented by a ground motion history.

These accelerograms are generally applied in the two planar directions to the bottom nodes, while the vertical component of the record is neglected.

The characteristics of the ground motion history significantly affect the response. Therefore, the EN 1998-1 suggests using at least seven time histories, and for the design it is used the average response parameter. Among the parameters that influence in the definition of the ground motion time history is the ground composition that directly affect the shear wave velocity: it is higher in rocky grounds and lower in muddy grounds. Other parameters that influence the ground motion selection are: fault distance and peak ground acceleration and velocity.

Once the ground motions are selected, they are manipulated: indeed, they are subjected to scaling and matching. In the first process each acceleration record is scaled to match a single period or a range of periods. During the spectrum matching, instead, the frequency content of records is handled to fit the elastic design spectrum provided by the governing code (i.e. NTC).

The parameter that helps the most in monitoring the dynamic response of a structure is the drift ratio: it is defined for the i -th floor as:

$$D_i = \frac{(disp_{(i)} - disp_{(i-1)})}{h} \quad (54)$$

Where h represents the vertical distance between two successive floors.

4.14- Optimization strategy for FRP design

The optimization procedure developed during this thesis is based on the TCL version of OpenSees and on a TCL script that manages the optimization cycle. The optimization algorithm aims to find the optimized number of FRP confined columns and their disposition, relying on the following condition:

$$\text{- Minimize } C = \sqrt{\sum_{i=1}^{N_S} (|D_i|^2)} \quad (55)$$

In eq.(55) the quantity expressed by the coefficient D represents the interstory drift ratio, taken in his absolute value and dimensionless.

Moreover, in addition to the objective function C a bond is added to find the optimized solution: the interstory drift ratio at each story cannot overcome a given value p .

The input conditions are based on the nonlinear analysis carried out for the bare RC structure: once that the nonlinear model is available a first static nonlinear analysis is performed to assess the ductility. Then the structure is tested through the dynamic nonlinear analysis to set the other conditions about the inter-story drift ratio. As described in the previous list, these parameters will act as terms of comparison for the optimization cycle.

Once that the described parameters are obtained, the FRP jackets are added to the columns. In OpenSees the FRP are implemented through the *ConfinedConcrete01* that allows to define all the FRP characteristics, such as its thickness, its elastic modulus and the ultimate state characteristics.

For first the FRP retrofit system is dimensioned: it has just to respect the condition of the effective confinement expressed by eq.(10): within this thesis it is assumed bigger than 0.07, as described by Lam & Teng on their “*Design-oriented stress–strain model for FRPconfined concrete in rectangular columns*”.

The FRP optimized sizing is not treated within this thesis because it aims to establish, for a given FRP thickness, the optimal layout of FRP-confined columns. Furthermore, a complete FRP confinement of RC elements is considered.

After dimensioning the FRP wraps, the optimization procedure gets to the heart. It is based on the definition of a Genetic Algorithm, whose workflow is summarized below:

- 1- Initialization
- 2- Genesis of a new population (offspring or child) from the combination of the individuals of previous generation
- 3- Execution of the analysis to assess the quality of each child/offspring
- 4- Local search on each child
- 5- The new population replaces the previous one
- 6- Back to step 2 for a new generation

During the initialization step the first population of parents is created: it consists of a subset of all the possible combinations of the twenty columns, based on the two possible materials available for each column: unconfined concrete or FRP-confined concrete. From the parental population, the new offspring forming the new population is generated. Each child is in the form of a binary string, formed by twenty elements: each element represents which material constitutes the column, 0 for unconfined concrete and 1 for FRP-confined concrete.

Two process are involved in the generation of the child population: crossover and mutation. Through the crossover technique the child is created from two random parents, while by mean of the mutation process some genes change to improve the quality of the child.

Through the tournament selection procedure, a child is randomly picked to run the analysis of the frame: the OpenSees output files produced through the recorder command are used by the GA to monitor the quality of each child.

At this point through the local search the previous population is improved: for each child, all the solutions close to him are inspected, in order to find a better genes configuration and a new improved population is generated. Now a new cycle is ready to be run again, starting with the generation of a new children population.

The GA functioning is described above, while the execution of the program will be described below.

The Genetic.exe must be executed within its folder through command line. The folder contains the TCL script that describe the RC frame, split into twelve different files, the two applications, the txt files for the ground motion history and the latest version of OpenSees.

The application counts of two different functioning modes:

- 1- The first one involves the first launch of the application: the first random population is created.
Genetic.exe <num_popolazione> <num_elite> <pressure> <outputfile>
When the execution is forced to stop, the results are saved in the outputfile
- 2- With the second mode, the program can be re-run from where it was interrupted
Genetic.exe <num_popolazione> <num_elite> <pressure> <inputfile> <outputfile>
The inputfile is a file generated in the previous run and is the basis for the new cycle. As in the first mode, the results are saved in the outputfile.

For instance, if the application is launched for the first time and we want to save the result in result.txt:

```
Genetic.exe <num_popolazione> <num_elite> <pressure> result.txt
```

The program can be stopped from command line in any moment by the combination: CTRL+C

After that the result.txt is created and it contained the population of the last generation carried out. Moreover, if in the current generation a better solution than all of those found up to that point was found, this will also be saved in the output file, i.e. result.txt.

Now, if it is necessary to resume the execution of the application from where it was interrupted, is enough to follow this command:

```
Genetic.exe <num_popolazione> <num_elite> <pressure> result.txt resultbis.txt
```

Where result.txt is the input file, while resultbis.txt is the new output file.

The three parameters to be inserted in the command line are:

- <num_popolazione>: it represents the number of population elements. The higher the <num_popolazione> the better it works, but the execution time suffers high values of this parameter.

- <num_ elite>: it represents the number of elements that, in a given generation, from the old population go directly to the new one. The elements are picked in descending order of fitness, starting from the best one.
- <pressure>: this parameter is used for the selection of two parents when a new child has to be generated. The higher the <pressure> the higher the probability of selecting the best parents is. It has to be noticed that the selection of the two best parents does not lead to the generation of the best child.

The following configuration are suggested:

- For running the analysis on personal computer: 10 1 3 respectively;
- For running the analysis on the server: 50 5 5 respectively

The application allows also to verify if a given solution has a good quality, acting according to the following steps:

- 1- Open the command line within the principal folder of the program;
- 2- Create a new folder giving it a name at will;
- 3- Insert the following command: `opensees 5MainFRPGen.tcl prova 0 1 0 1 ... 1 0`, where 0/1 indicates the configuration to test
- 4- Click Enter to run the analysis

The specific configuration is tested through a single analysis with OpenSees. The output files are stored in the folder created in the step 2, and they can be checked to verify their effective quality. The results of this analysis must coincide with those produced during the optimization cycle.

5. CASE STUDY

In this thesis a five-story single bay reinforced concrete frame is investigated. It has been designed to represent a typical non-ductile residential building: in fact, it is characterized by poor material properties, violation of capacity design standard and inadequacy of steel reinforcement, in particular for what concerns the transversal reinforcement.

As described in the methodology chapter, more than one frame has been modelled before modelling the five story one. Moreover, the frames are built both in OpenSees and in SAP2000, but the aim of this thesis is to investigate the response of the five-story frame, whose OpenSees modelling is described below.

5.1- Materials

The concrete of class C25/30 is used for concrete elements: it shows a cylindrical compressive strength of 25 MPa. The steel used for longitudinal and transversal reinforcement is of class B450C with a nominal strength of 450 MPa.

5.2- Geometry

The reinforced concrete frame consists of five story and one bay in both directions (Fig. 20). The story height is of 3.6 m and the bay has a width of 6.0 m. The elements are grouped in two groups in according to their functions: columns are the vertical elements while beams are the horizontal one.



Fig. 20 Five story RC frame views: a) 3D view; b) 2D view

b)

The first one has a cross section of 0.35x0.35 m, constant along the height; the beams have a rectangular section of 0.35x0.55 m. The longitudinal bars in the columns are six with a diameter of 20.0 mm; the ties for transversal reinforcement have a diameter of 8.0 mm with a spacing of 250mm. The beams are reinforced with ten longitudinal bars with a diameter of 24.0 mm and with ties of 8.0 mm of diameter, with a spacing of 250.0 mm (Fig. 21).

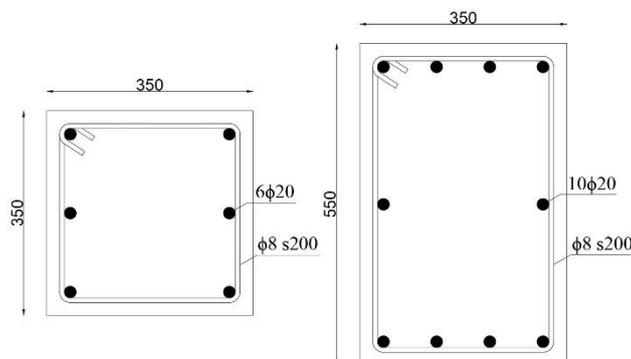


Fig. 21 Beam and column cross-section

The use of FRP sheets has been designed in according with just one constraints, the one provided by eq.(10), regarding the effective confinement. For this reason, were chosen the FRP sheets with an elastic modulus of

460.0 GPa and with a total thickness of 1.0 mm: this provide a value of c_{eff} equal to 0.074, that is greater than 0.07 (

| b [mm] | h [mm] | f_{ck} [Mpa] | f_{yk} [Mpa] | E_f [Mpa] | t_f [mm] | f_l [N/mm2] | c_{eff} [-] |
|------------------|------------------|--------------------------------|--------------------------------|-------------------------------|------------------------------|---------------------------------|-------------------------------|
| 350 | 350 | 25 | 450 | 460000 | 1 | 1.84 | 0.074 |

Table 3).

| b [mm] | h [mm] | f_{ck} [Mpa] | f_{yk} [Mpa] | E_f [Mpa] | t_f [mm] | f_l [N/mm2] | c_{eff} [-] |
|------------------|------------------|--------------------------------|--------------------------------|-------------------------------|------------------------------|---------------------------------|-------------------------------|
| 350 | 350 | 25 | 450 | 460000 | 1 | 1.84 | 0.074 |

Table 3 Calculation of effective confinement c_{eff}

For the determination of c_{eff} the smoothing of right angle has been considered, as it is suggested in practice: the International codes suggest a minimum value of the corner radius R_C equal to 20mm to improve the effectiveness of the confinement. Therefore, R_C is taken equal to 20mm.

To each element of the frame an identification number has been assigned, in particular the columns have an ID number from 1 to 20, ordered from the bottom to the top (Fig. 22).

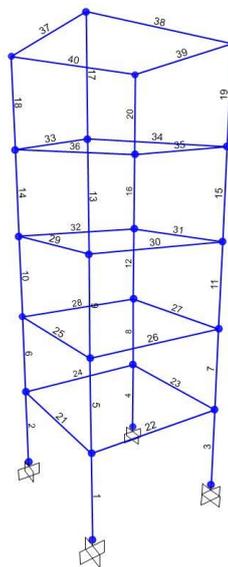


Fig. 22 Elements ID number

5.3- Static Nonlinear Analysis

To perform the pushover analysis, the static distribution force is chosen and so it is necessary to determine the Design Spectrum. The frame is supposed to be constructed in Rome, on the flat ground of class B.

According to eq.(41) the behavior factor is obtained, as shown below:

- $q_0 = 4.5 \frac{\alpha_u}{\alpha_i}$ for RC frame
- $\frac{\alpha_u}{\alpha_i} = 1.2$ for RC frame with more than one story and with one bay
- $K_R = 1$ for regular frame

Therefore, the maximum acceptable value for q is 5.4, but for the determination of the DS it was taken conservatively equal to 3.

Moreover, a damping ratio equal to 0.05 it was considered, that is an usual value for RC frame.

The DS is provided by OPENSIGNAL (Cimellaro and Marasco 2015), after setting all the parameters described in chapter 4.6-. The Fig. 23 and Fig. 24 show the two versions of the Design Spectrum: the first one is for elastic building, while the second one is for dissipative building.

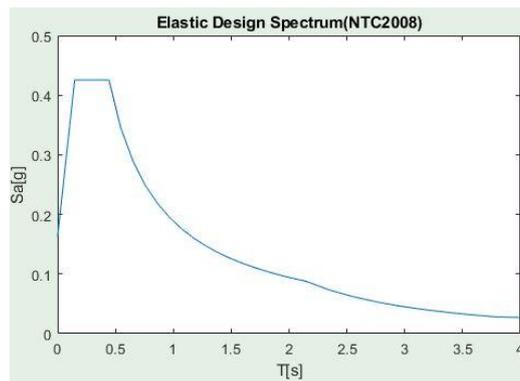


Fig. 23 Elastic Design Spectrum

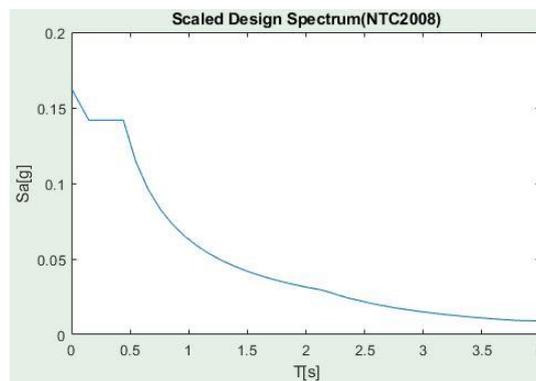


Fig. 24 Scaled Design Spectrum

From eq.(52) is determined the value of the fundamental period T_1 , where the maximum lateral displacement d obtained from the load combination eq.(53) is equal to 0.161 m:

$$T_1 = 2 \cdot (0.161)^{0.5} = 0.802s \quad (56)$$

From the DS and from the value of the calculated fundamental period the spectral acceleration $S_d(T_1)$ used to determine the static forces is:

$$S_d(T_1) = 0.201m / s^2 \quad (57)$$

According to eq.(51) F_h is equal to:

$$F_h = S_d(T_1)W\lambda / g \quad (58)$$

Now it is possible to determine the static forces to be applied at each story (Table 4)

| Story | z [m] | F _h [N] | W [N] | F _i [N] |
|-------|-------|--------------------|-----------|--------------------|
| 1 | 3.6 | 77360.82 | 759542.43 | 5207.77 |
| 2 | 7.2 | 77360.82 | 759542.43 | 10415.53 |
| 3 | 10.8 | 77360.82 | 759542.43 | 15623.30 |
| 4 | 14.4 | 77360.82 | 759542.43 | 20831.06 |
| 5 | 18.0 | 77360.82 | 737500.38 | 25283.17 |

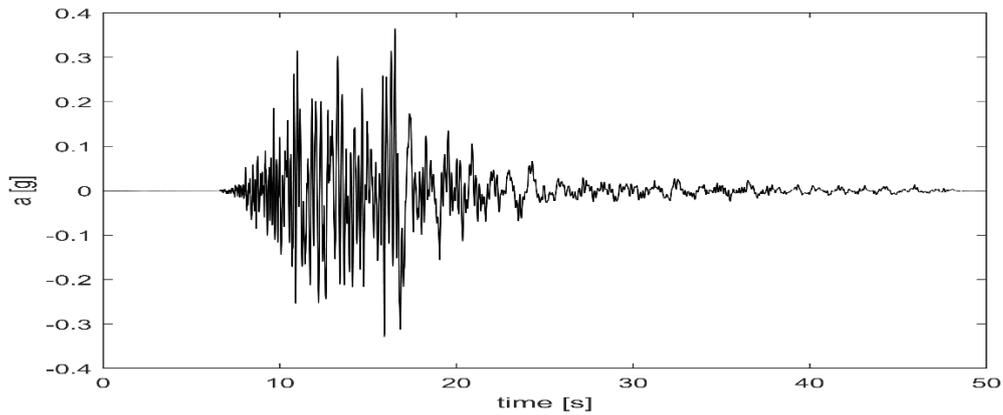
Table 4 Determination of static forces

It can be noticed that the distribution of the static forces has an inverse triangular shape, with the maximum magnitude in correspondence of the last story, following the first modal shape.

5.4- Dynamic Nonlinear Analysis

The nonlinear time history analysis is based on seven accelerograms for design purposes. In this case, since the final aim of this thesis is not the design, just one accelerogram will be used. In particular the Norcia 2016 earthquake ground motion is chosen from the PEER database, following the criteria listed below:

Using OPENSIGNAL the accelerogram is filtered and scaled according with the DS, and it assumes the below shape:



Once that the accelerogram is available, it is applied at the structure in both horizontal directions: since it is dimensionless, for the analysis a factor equal to gravity acceleration is applied.

5.5- FINITE ELEMENT MODEL

Before starting with the analysis, a sufficiently accurate finite element model is built. To obtain such a model, the structure is modelled through different steps both in SAP2000 (CSI 2002) and in OpenSees (McKenna 2011). As described in the methodology chapter, elastic models are first modelled and compared through the eigenvalue analysis. Then, non-linearities are introduced, for what concerns materials and elements. The two models are then subjected to non-linear analysis: static and dynamic.

Two different software are used to enhance the credibility of the OpenSees model and also to compare its knowledge.

5.6- Geometry

The structure consists of two principal frames connected by beams. Both in SAP2000 and in OpenSees rigid diaphragms were created at each floor, constraining each floor to work as a rigid body, neglecting displacements and rotations out of the plane. At the bottom nodes all the six DOFs are restrained, giving the characteristics of perfect joint. In OpenSees through the *rigidDiaphragm* command the external joints at each story (named *slave* nodes) are forced to move as if in a rigid plane with the centre of mass (named *master* node). To make in agreement both software, the plastic hinge length was taken equal to the 5% of the considered concrete element.

5.7- Materials

In both software concrete materials were firstly introduced. In SAP2000 several concrete types can be selected from a list, in which they are distinguished in according to the specific standards. From the Italian code NTC2008 the concrete class C25/30 is picked.

There are many models that describe the stress-strain relationship for some concrete types or provided by publications, such as concrete with zero tensile strength, confined concrete, concrete with linear tension

softening, ecc. In OpenSees each concrete type is defined twice to consider the differences between the one used inside and the one used outside the area confined by stirrups. Both software provides for concrete the tensile strength. In particular, for the section not retrofitted the *Concrete02* material is used.

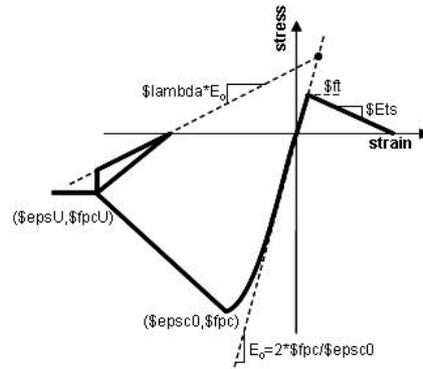


Fig. 25 Concrete02 stress-strain relationship - opensees.berkeley.edu

The longitudinal and transversal reinforcement are modelled using bilinear steel model with hardening, that in OpenSees is identified through the *Steel02* command.

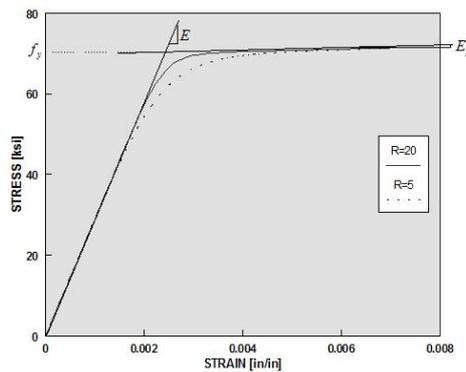


Fig. 26 Steel02 stress-strain relationship - Opensees.berkeley.edu

In OpenSees there is also a particular concrete material, called *ConfinedConcrete01*, that consider the confinement offered by transversal stirrups together with FRP jacketing. The formulation of this material is provided by (Braga, Gigliotti et al. 2006). In this case, in according with (McKenna 2011) “the confined concrete has not tensile strength and degraded linear unloading/reloading stiffness as proposed by (Karsan and Jirsa 1969). The BGL (i.e. Braga, Gigliotti, Laterza) model accounts for confinement effects due to different arrangements of transverse reinforcement and/or external strengthening such as steel jackets or FRP wraps.

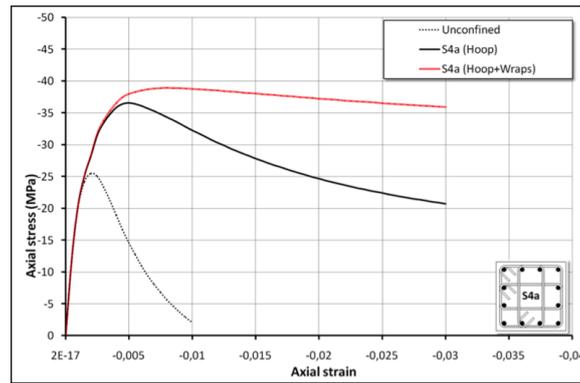


Fig. 27 Stress-strain relationship for square section with and without FRP - (McKenna 2011)

The confinement effect along the column is described as well. In order to obtain the compressive envelope, curve a non-linear approach is performed at each increment of column axial strain. The sought curve is obtained crossing different stress-strain relationships, each of which corresponding to a different level of confinement”.

5.8- Elements

In SAP2000 the elements are defined giving the coordinates of two points and assigning the specific section defined before. The non-linear model stands out from the elastic one for the presence of the steel reinforcement within concrete element and the plastic hinges. Each beam-column element has six degrees-of-freedom.

In OpenSees the differences between elastic and inelastic elements are not negligible. When defining an elastic beam-column element is enough to model and assign the *elasticBeamColumn* element: for three-dimensional problems, it is defined through the elastic modulus of concrete, the shear modulus, the cross-section area, the torsional moment of inertia of cross section and the two second moment of area about z-axis and y-axis. It is important to note that for elastic model no material must be defined, since the calculation of the stiffness is based just on the integration of Young modulus along the element length. Instead, when dealing with inelastic model, after defining each material with its stress-strain relationship, the nonlinear elements must be defined. Through the *forceBeamColumn* element type with Hinge-Radau integration method, the nonlinear elements are defined. This element type is particularly suitable for concentrated plasticity approach, confining the rotational springs at the element extremities.

In accordance with SAP2000, the plastic hinge length is set equal to the 5% of the element length, from both the element ends. The interior section type can be elastic or inelastic: as it will be shown below the results are more consistent with SAP2000 using a nonlinear section also for the element interior region.

6. RESULTS

6.1- Eigenvalue analysis

To test the effectiveness of the OpenSees model, several eigenvalue analyses were carried out on five different elastic frames, from one up to five story.

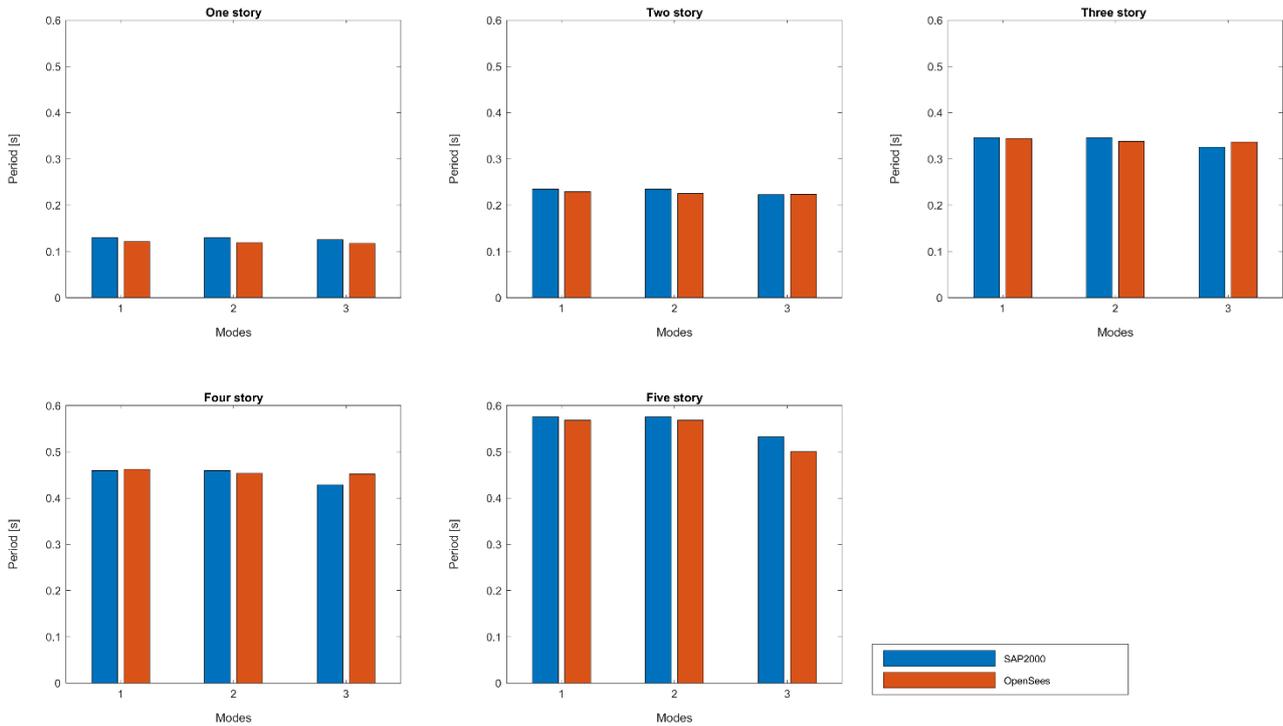


Fig. 28 Comparison of modal periods for Elastic frame

As shown in Fig. 28 the elastic models built in SAP2000 and in OpenSees are in agreement for what concerning the first three periods obtained from the eigenvalue analysis.

The eigenvalue analysis performed on the nonlinear model with unconfined concrete and on the model with FRP-confined concrete have shown that the modal periods are the same (Fig. 29). This because the initial stiffness of FRP-confined concrete material is independent of the thickness of FRP wrap. As mentioned in the previous chapter, FRP provides a passive confinement, so its contribution is evident in the post elastic concrete behavior.

OpenSees uses, for modal analysis, the elastic stiffness that is the same for all the FRP-confined concrete stress-strain curves produced for the same concrete material but varying the FRP stiffness.

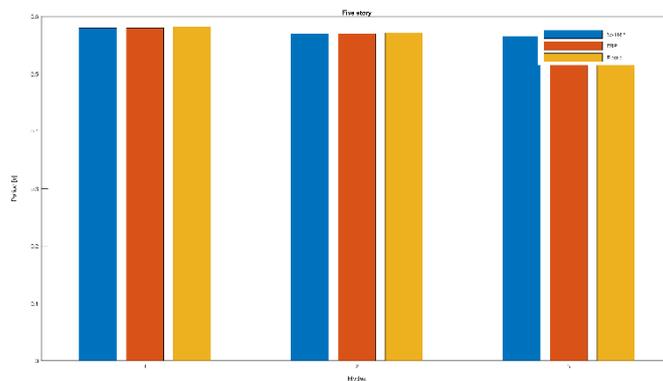


Fig. 29 Fundamental periods comparison

The fiber section used to assign the elements defines the stiffness at each analysis step from integration of the stiffnesses of each fiber section. And the fiber section defines the stiffness from integration of the Young modulus of each material point of the section. In the stiffness of the section there are contributions from both concrete and steel materials.

Moreover, the introduction of steel reinforcement is reflected in the slight decrease of fundamental periods, thus making the structure stiffer.

6.2- Static nonlinear analysis

The capacity curve shown in Fig. 30 is obtained through the procedure described in the previous chapter, for the inverse triangular distribution of static forces. Each force has been applied in the centre of the mass of each story

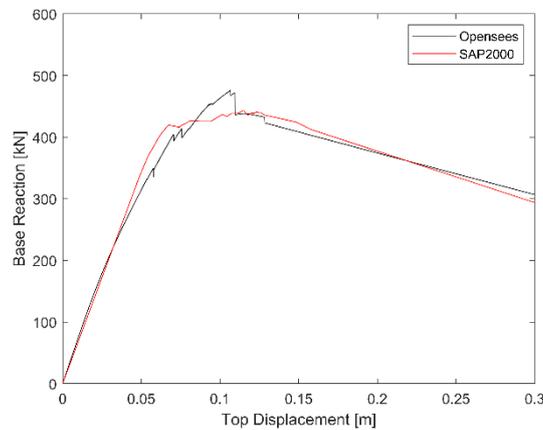


Fig. 30 Capacity curve for five story RC frame

Both software provides the same response of the structure subject to the nonlinear static analysis: in particular they highlight a first elastic portion, then a plastic portion with a plateau more evident in SAP2000 and then a softening portion.

The pushover analysis performed on the five story RC frame had the aim to compare the two non-linear models, but it has not been inserted in the optimized solution algorithm, because the time history analysis was preferred.

6.3- Dynamic nonlinear analysis

The Nonlinear Time History Analysis (NTHA) is carried out first for the bare frame: all the frames from one up to five story were subjected to NTHA to compare the results between SAP2000 and OpenSees. In Fig. 31 and in Fig. 32 the comparison between the two software in terms of story displacement is shown, for both planar directions for the five story RC frame.

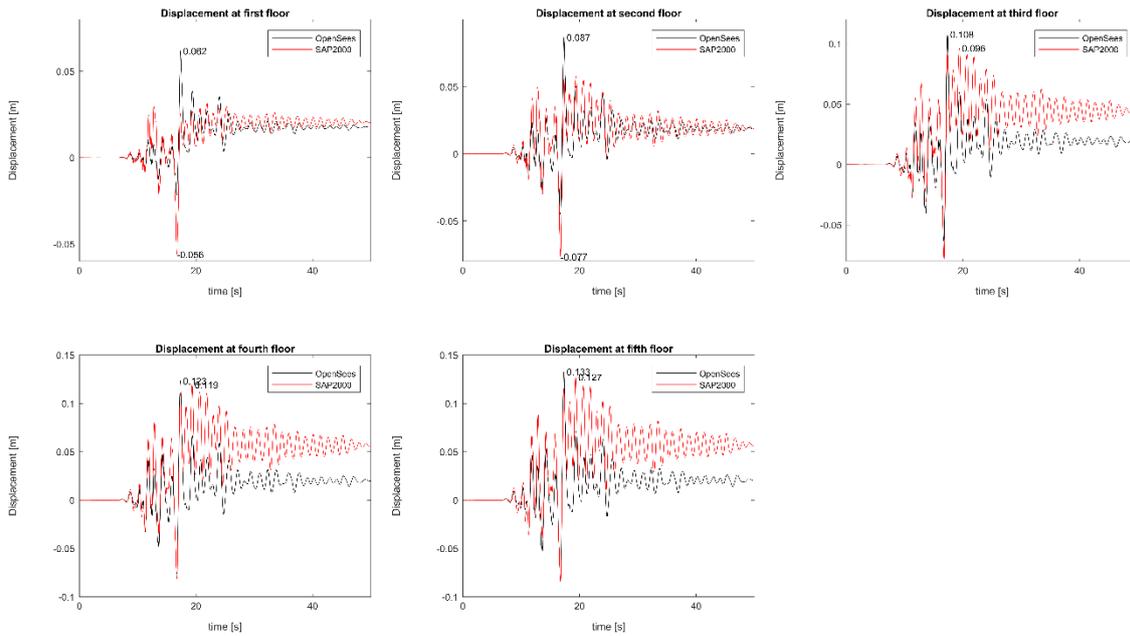


Fig. 31 Displacement comparison between SAP2000 and OpenSees for direction 1

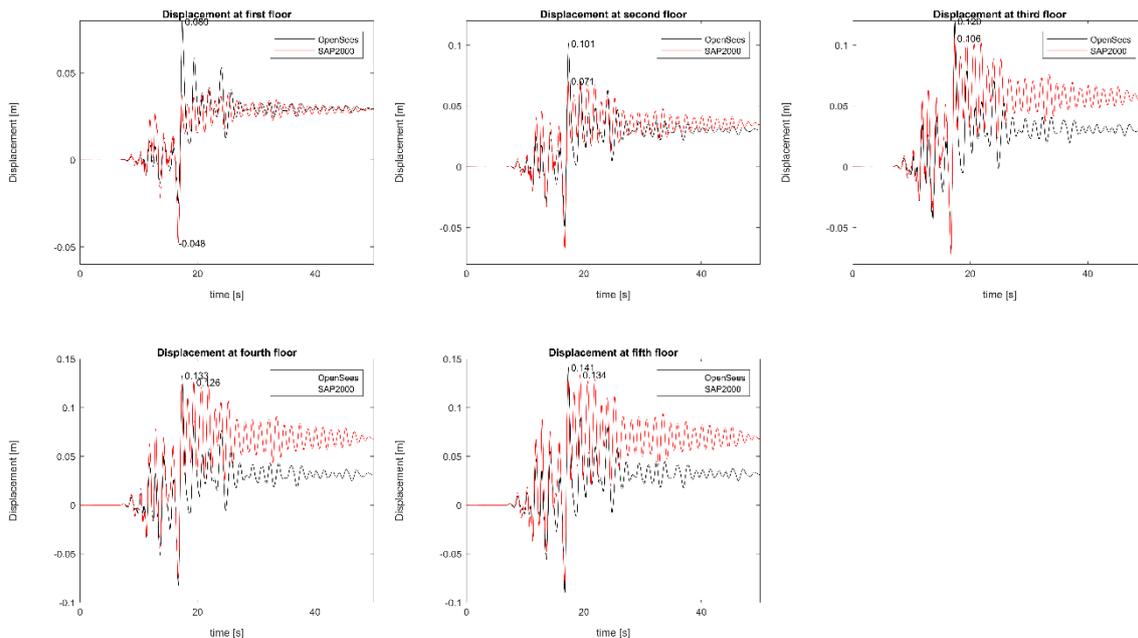


Fig. 32 Displacement comparison between SAP2000 and OpenSees for direction 3

From the two images is possible to note that the overall structural behavior is similar for both software: in particular a first elastic portion, highlighted by a moving average equal to zero (during the time interval 0-17 s), is followed by an inelastic phase in which the moving average increases. Moreover, both software shows a residual deformation at the end of the seismic record, more evident in SAP2000 for the last three story.

The differences between the two software for the inelastic response are probably due to the different definition of the nonlinear elements and of the plastic hinges, but the elastic behavior and also the peaks in

terms of displacements are in good agreement. For this reason, the NTHA results obtained from OpenSees are used for the definition of the optimization strategy.

The main parameters acquired from NTHA are the interstory drift ratios: for the bare RC frame they are represented in the following Fig. 33 Fig. 34 for both directions.

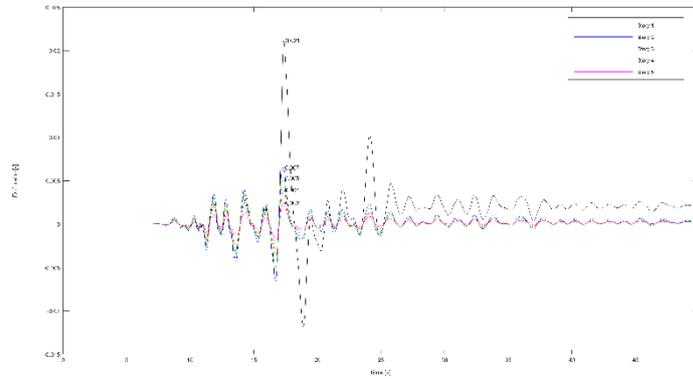


Fig. 33 Interstory Drift ratio – dir. 1

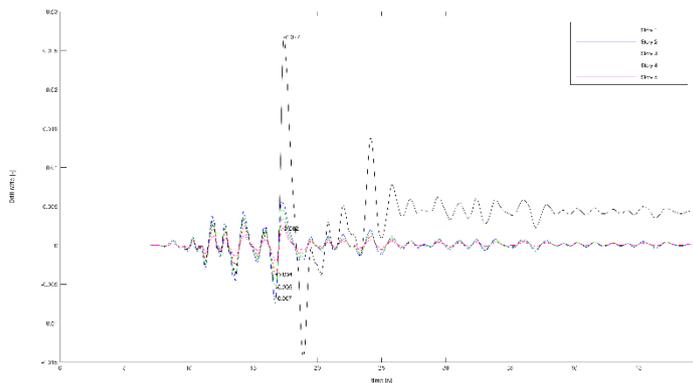


Fig. 34 Interstory Drift ratio – dir. 3

As it is possible to see from the last two figures, the maximum values of interstory drift ratio involve the first story of the frame for both directions and they are summarized in the below TABLE.

| Story | D1 [%] | D3 [%] |
|-------|--------|--------|
| 1 | 2.1 | 2.7 |
| 2 | 0.7 | 0.7 |
| 3 | 0.5 | 0.5 |
| 4 | 0.4 | 0.4 |
| 5 | 0.2 | 0.2 |

Table 5 Interstory drift ratio

6.4- Optimization

From the results of NTHA summarized on Table 5 the two coefficients for the calculation of the objective function are obtained:

$$C_1 = 0.021 \quad (59)$$

$$C_3 = 0.029 \quad (60)$$

where the two subscripts used to distinguish the two coefficients refer to the considered earthquake direction. Thus, the optimization procedure is focused on the following objective function:

$$C = \sqrt{\sum_{i=1}^{N_s} (|D_i|^2)} < 0.029 \quad (61)$$

The additional bond for the optimization research is represented by the coefficient p : since the interstory drift ratios for the story from two to five are lower than 1.0%, the maximum acceptable value for p is 0.7%.

Once that the two parameters are set, the optimization analysis can be performed. At each iteration, starting from a random combination of wrapped columns, the program adds and/or removes and/or moves a random number of FRP-confined columns, in order to achieve the optimal layout, respecting the objective function and following the principles of the Genetic Algorithm described in the previous chapters. The quality of each iteration is automatically assessed through the NTHA.

In Fig. 35 the comparison between the displacement for bare frame and for retrofitted frame is displayed: it is possible to noticed that the elastic behavior is almost the same between the two situations, while a more evident plasticization involved the non-retrofitted frame, causing greater displacements and therefore also greater deformations. Here the results for the first story are reported, but the same decrease, although in minor terms, affected also the upper story, as is understandable from the following figures.

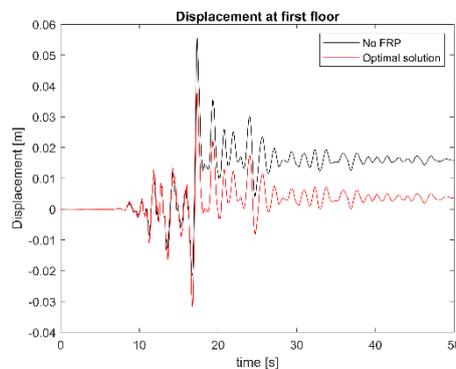


Fig. 35 Displacements comparison dir. 1

In Fig. 36 and in Fig. 37 the interstory drift ratio history for both directions are shown: it can be noticed that the aim of the optimization analysis is achieved, in fact, with respect to Fig. 33, the value of maximum

interstory drift ratio is significantly decreased passing by 2.1% to 1.1% for direction 1 and from 2.7% to 1.3% for direction 3. Moreover, also the residual deformation which interested the bare frame is almost negligible for the FRP-retrofitted frame.

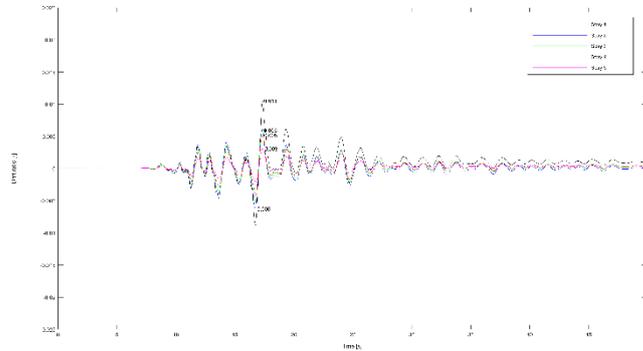


Fig. 36 Interstory drift ratio for optimal solution – dir.1

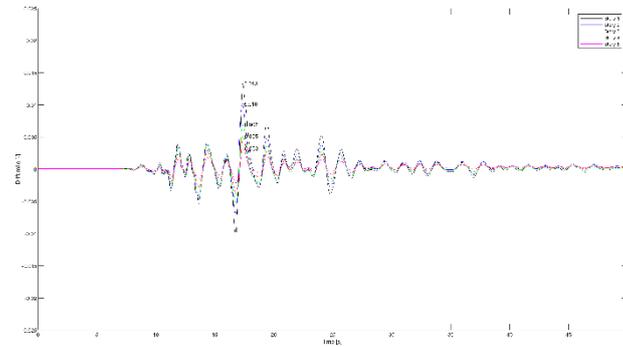


Fig. 37 Interstory drift ratio for optimal solution – dir.3

In Fig. 38 all the solutions found by the GA procedure are shown. It can be noticed that two are the minimum solutions, but just the one corresponding to seven FRP-wrapped columns is admissible, because it respects both the objective function and the bond. In Fig. 39 are shown just the admissible solutions.

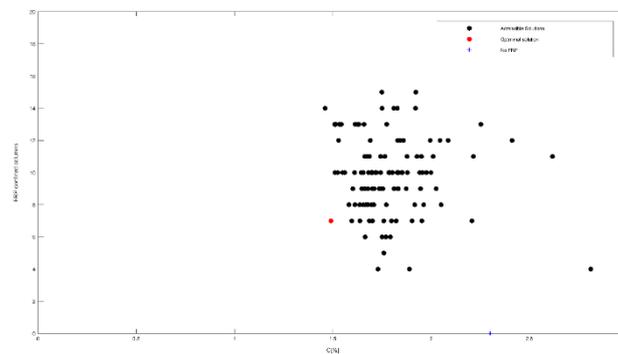


Fig. 38 GA solutions

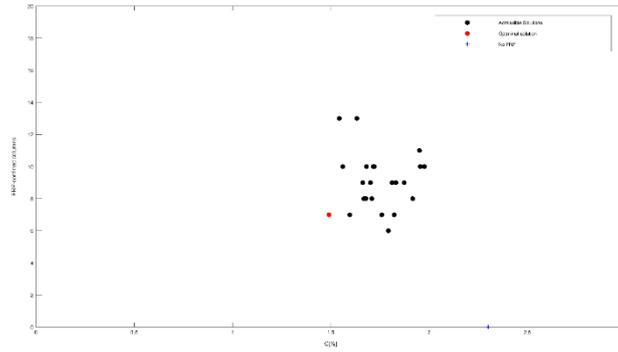


Fig. 39 Admissible solutions

It is possible to noticed that an evident relation between the number of FRP-confined columns is not distinguishable, the most important factor that affects the optimization research is the position of the confined columns (Fig. 40).

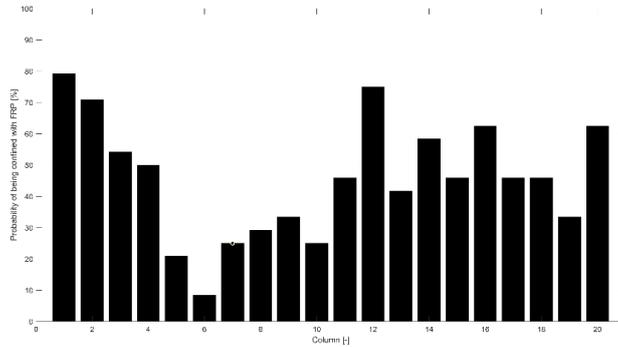


Fig. 40 Probability of columns of being confined with FRP during the optimization algorithm

In Fig. 40 is highlighted the probability of each of the twenty columns of being wrapped with the FRP sheets, as results from the optimization analysis. The optimization analysis has shown that the columns that link the ground floor with the first floor (column 1 to 4) have a probability greater than 50% of being wrapped: this makes sense because, as shown in Fig. 33 and in Fig. 34, the interstory drift ratio, resulting from the NTHA, has its maximum in correspondence with the first story.

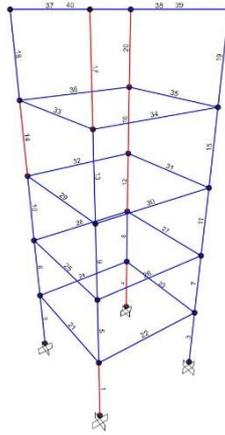


Fig. 41 Optimal disposition of FRP confined columns

The optimal solution is shown in Fig. 41 (also shown in Fig. 38 and Fig. 39 through the red dot): in red are represented the FRP-confined columns, while in blue there are the reinforced concrete elements. In this solution the columns 1, 4, 12, 14, 16, 17 and 20 are wrapped: as is shown in Fig. 40, except for column 17 all the other columns have a probability of being wrapped greater than 50%; moreover, column 1 and 12 are the ones with the higher value of the probability (more than 75%) and column 20 has a probability greater than 60%.

It should be noted that the Fig. 40 is useful to know what are the columns most interested by the FRP wrapping during the optimization procedure, but it is not sufficient to pick the ones with the greater value of probability, it is necessary to compose the right combination among all of them.

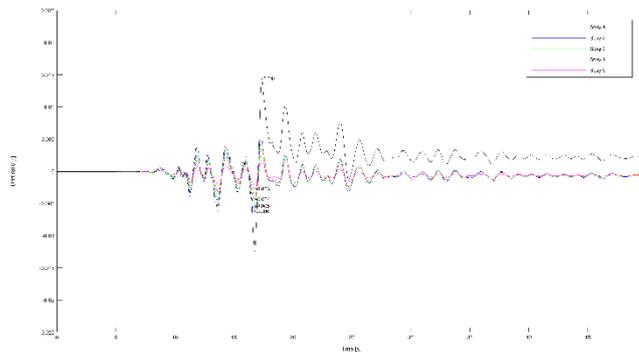


Fig. 42 Interstory drift ratio for columns most interested by FRP wrapping

In Fig. 42 is shown the interstory drift ratio in direction 1, obtained by wrapping with FRP sheets the columns 1, 2, 12, 14, 16 and 20, i.e. those that have been affected by the FRP jacketing in at least 60% of the simulations. As can be seen from the comparison between Fig. 42 and Fig. 36 the solution presented in the first figure is worse than the optimal one, described in the second figure.

7. CONCLUSIONS

During this thesis the behavior of a RC frame was studied. The high values of the interstory drift ratio, in particular in correspondence of the first story, required a strengthening strategy, which was identified with the use of FRP-wraps. The FRP systems were chosen to simulate a retrofit intervention on an existing building, on which was not possible to intervene with the usual technologies, such as shear walls or base isolators.

The purpose of finding the optimal use of FRP on the columns has been achieved through the definition of a Genetic Algorithm, an iterative biology-inspired procedure able to evolve and mutate itself in order of finding the optimal solution. Each iteration has been tested through a NTHA performed on the five story frame and carried out using the open source software OpenSees.

This procedure has allowed of finding the best solution, that is characterized by seven FRP-confined columns, that leads to a significant reduction of the maximum interstory drift ratio and at the same time leaves almost unchanged the others. Moreover, as shown in Fig. 40, it was possible to establish on which columns is better to focus for the FRP-wrapping, looking for the optimal combination.

The presented procedure within this Master Thesis could be applied in the future on a greater number of buildings, in order to be able to define a general methodology for the use of FRP, that can be useful in the during the design of the retrofit intervention.

8. APPENDIX A: OPENSEES CODE (TCL VERSION)

```
# Start of model generation
# Create ModelBuilder
# -----
model BasicBuilder -ndm 3 -ndf 6

# Define geometry
# -----
# ----- frame configuration
set NStory 5;
set NBay 1;
set NBayZ 1;
set NFrame [expr $NBayZ + 1];

# define GEOMETRY

set LCol 3.6; #[m]
set LBeam 6.0; # [m]
set LGird 6.0;

# define NODAL COORDINATES
set Dlevel 10000;
set Dframe 100;
for {set frame 1} {$frame <=[expr $NFrame]} {incr frame 1} {
    set Z [expr ($frame-1)*$LGird];
    for {set level 1} {$level <=[expr $NStory+1]} {incr level 1} {
        set Y [expr ($level-1)*$LCol];
        for {set pier 1} {$pier <= [expr $NBay+1]} {incr pier 1} {
            set X [expr ($pier-1)*$LBeam];
            set nodeID [expr $level*$Dlevel+$frame*$Dframe+$pier]
            node $nodeID $X $Y $Z;
        }
    }
}
}
```

```

# Define material(s)
# -----
set steelRebar1 1
set steelRebar2 12
set steelRebar345 11

set concCover1 2
set concCore1 3
set concCover2 222
set concCore2 322
set concCover345 21
set concCore345 31

# STEEL B450C
set Fy 450000000.0
set fu 525000000.0
set E0 210000000000.0
set Bs1 0.01; #-0.01
set Bs2 0.01; #-0.01
set Bs345 0.01; #-0.01
uniaxialMaterial Steel02 $steelRebar1 $Fy $E0 $Bs1 10. 0.925 0.15;
uniaxialMaterial Steel02 $steelRebar2 $Fy $E0 $Bs2 10. 0.925 0.15;
uniaxialMaterial Steel02 $steelRebar345 $Fy $E0 $Bs345 10. 0.925 0.15;

# CONCRETE C25/30
set Ec 31480000000.0
set fc -25000000.0
set epsc01 [expr -1.613e-03]
set epsc02 [expr -1.613e-03]
set epsc0345 [expr -1.313e-03]
set fcu -3000000.0
set epsU1 -0.005;
set epsU2 -0.005;
set epsU345 -0.005;

```

```

set lambda 0.1
set ft 2200000.0
set Ets 2000000000.0

uniaxialMaterial Concrete01 $concCore1 $fc $epsc01 $fcu $epsU1
uniaxialMaterial Concrete01 $concCover1 $fc $epsc01 $fcu $epsU1

uniaxialMaterial Concrete01 $concCore2 $fc $epsc02 $fcu $epsU2
uniaxialMaterial Concrete01 $concCover2 $fc $epsc02 $fcu $epsU2

uniaxialMaterial Concrete01 $concCore345 $fc $epsc0345 $fcu $epsU345
uniaxialMaterial Concrete01 $concCover345 $fc $epsc0345 $fcu $epsU345

set FRP1 48;
set FRP2 47;
set FRP345 46;

uniaxialMaterial ConfinedConcrete01 $FRP1 S1 $fc $Ec -epscu -0.0035
-nu 0.2 0.35 0.008 0.15 $Fy $E0 0.0 1000.0 0.02 -wrap 0.03 0.0005 0.1
4200000000.0 690000000000.0 -stRatio 0.85

uniaxialMaterial ConfinedConcrete01 $FRP2 S1 $fc $Ec -epscu -0.0035
-nu 0.2 0.35 0.008 0.15 $Fy $E0 0.0 1000.0 0.02 -wrap 0.03 0.0005 0.1
4200000000.0 690000000000.0 -stRatio 0.85

uniaxialMaterial ConfinedConcrete01 $FRP345 S1 $fc $Ec -epscu -0.0035
-nu 0.2 0.35 0.008 0.15 $Fy $E0 0.0 1000.0 0.02 -wrap 0.03 0.0005 0.1
4200000000.0 690000000000.0 -stRatio 0.85

# Define single point constraints
# -----
fixY 0.0 1 1 1 1 1 1;
# Define section(s)
# -----
set ColSecTag1 1;
set BeamSecTag1 2;
set GirdSecTag1 3;
set IdColTors1 33;
set TColSecTag1 44;

```

```
set ColSecTag2 12;  
set BeamSecTag2 27;  
set GirdSecTag2 37;  
set IdColTors2 337;  
set TColSecTag2 447;
```

```
set ColSecTag345 112;  
set BeamSecTag345 212;  
set GirdSecTag345 312;  
set IdColTors345 3312;  
set TColSecTag345 4412;
```

```
set EColSecTag 11;  
set EBeamSecTag 22;  
set EGirdSecTag 33;
```

```
set FRPIdColTors1 313;  
set TFRPColSecTag1 444;
```

```
set FRPIdColTors2 213;  
set TFRPColSecTag2 244;
```

```
set FRPIdColTors345 513;  
set TFRPColSecTag345 544;
```

```
set HsecC 0.35;  
set BsecC 0.35;  
set coverC 0.03;  
set nfC 6;  
set strisce 3;  
set nBarsC [expr $nfC/$strisce];  
set fiC 0.02;  
set barAreaC [expr ($nfC*(3.14159*pow($fiC,2)/4))];
```

```

set Area1S [expr $barAreaC/$strisce];
set coverYC [expr $HsecC/2.0];
set coverZC [expr $BsecC/2.0];
set coreYC [expr $coverYC-$coverC];
set coreZC [expr $coverZC-$coverC];
set nfcoverYC 16;
set nfcoverZC 4;
set nfcoreYC 16;
set nfcoreZC 4;

set FRPColSecTag1 333;
section Fiber $FRPColSecTag1 {
#Define the core patch
patch quad $FRP1 $nfcoreZC $nfcoreYC -$coreYC $coreZC -$coreYC -$coreZC $coreYC
-$coreZC $coreYC $coreZC
# Define the four cover patches
patch quad $FRP1 2 $nfcoverYC -$coverYC $coverZC -$coreYC $coreZC $coreYC
$coreZC $coverYC $coverZC
patch quad $FRP1 2 $nfcoverYC -$coreYC -$coreZC -$coverYC -$coverZC $coverYC -
$coreZC $coreYC -$coreZC
patch quad $FRP1 $nfcoverZC 2 -$coverYC $coverZC -$coverYC -$coverZC -$coreYC -
$coreZC -$coreYC $coreZC
patch quad $FRP1 $nfcoverZC 2 $coreYC $coreZC $coreYC -$coreZC $coverYC -
$coreZC $coverYC $coverZC
# define reinforcing layers
layer straight $steelRebar1 $nBarsC $Area1S $coreYC $coreZC $coreYC -$coreZC;
layer straight $steelRebar1 $nBarsC $Area1S 0.0 $coreZC 0.0 -$coreZC;
layer straight $steelRebar1 $nBarsC $Area1S -$coreYC $coreZC -$coreYC -
$coreZC;
}

set FRPColSecTag2 233;
section Fiber $FRPColSecTag2 {
#Define the core patch

```

```

patch quad $FRP2 $nfcoreZC $nfcoreYC -$scoreYC $scoreZC -$scoreYC -$scoreZC $scoreYC
-$scoreZC $scoreYC $scoreZC

# Define the four cover patches

patch quad $FRP2 2 $nfcoverYC -$coverYC $coverZC -$scoreYC $scoreZC $scoreYC
$scoreZC $coverYC $coverZC

patch quad $FRP2 2 $nfcoverYC -$scoreYC -$scoreZC -$coverYC -$coverZC $coverYC -
$coverZC $scoreYC -$scoreZC

patch quad $FRP2 $nfcoverZC 2 -$coverYC $coverZC -$coverYC -$coverZC -$scoreYC -
$scoreZC -$scoreYC $scoreZC

patch quad $FRP2 $nfcoverZC 2 $scoreYC $scoreZC $scoreYC -$scoreZC $coverYC -
$coverZC $coverYC $coverZC

# define reinforcing layers

layer straight $steelRebar2 $nBarsC $Area1S $scoreYC $scoreZC $scoreYC -$scoreZC;
layer straight $steelRebar2 $nBarsC $Area1S 0.0 $scoreZC 0.0 -$scoreZC;
layer straight $steelRebar2 $nBarsC $Area1S -$scoreYC $scoreZC -$scoreYC -
$scoreZC;
}

set FRPColSecTag345 133;
section Fiber $FRPColSecTag345 {
#Define the core patch

patch quad $FRP345 $nfcoreZC $nfcoreYC -$scoreYC $scoreZC -$scoreYC -$scoreZC
$scoreYC -$scoreZC $scoreYC $scoreZC

# Define the four cover patches

patch quad $FRP345 2 $nfcoverYC -$coverYC $coverZC -$scoreYC $scoreZC $scoreYC
$scoreZC $coverYC $coverZC

patch quad $FRP345 2 $nfcoverYC -$scoreYC -$scoreZC -$coverYC -$coverZC $coverYC
-$coverZC $scoreYC -$scoreZC

patch quad $FRP345 $nfcoverZC 2 -$coverYC $coverZC -$coverYC -$coverZC -$scoreYC
-$scoreZC -$scoreYC $scoreZC

patch quad $FRP345 $nfcoverZC 2 $scoreYC $scoreZC $scoreYC -$scoreZC $coverYC -
$coverZC $coverYC $coverZC

# define reinforcing layers

layer straight $steelRebar345 $nBarsC $Area1S $scoreYC $scoreZC $scoreYC -$scoreZC;
layer straight $steelRebar345 $nBarsC $Area1S 0.0 $scoreZC 0.0 -$scoreZC;

layer straight $steelRebar345 $nBarsC $Area1S -$scoreYC $scoreZC -$scoreYC -
$scoreZC;
}

```

```

section Fiber $ColSecTag1 {
# Define the core patch
patch quad $concCore1 $nfcoreZC $nfcoreYC -$scoreYC $scoreZC -$scoreYC -$scoreZC
$scoreYC -$scoreZC $scoreYC $scoreZC
# Define the four cover patches
patch quad $concCover1 2 $nfcoverYC -$coverYC $coverZC -$scoreYC $scoreZC $scoreYC
$scoreZC $coverYC $coverZC
patch quad $concCover1 2 $nfcoverYC -$scoreYC -$scoreZC -$coverYC -$coverZC
$coverYC -$coverZC $scoreYC -$scoreZC
patch quad $concCover1 $nfcoverZC 2 -$coverYC $coverZC -$coverYC -$coverZC -
$scoreYC -$scoreZC -$scoreYC $scoreZC
patch quad $concCover1 $nfcoverZC 2 $scoreYC $scoreZC $scoreYC -$scoreZC $coverYC -
$coverZC $coverYC $coverZC
# define reinforcing layers
layer straight $steelRebar1 $nBarsC $Area1S $scoreYC $scoreZC $scoreYC -$scoreZC;
layer straight $steelRebar1 $nBarsC $Area1S 0.0 $scoreZC 0.0 -$scoreZC;
layer straight $steelRebar1 $nBarsC $Area1S -$scoreYC $scoreZC -$scoreYC -
$scoreZC;
}

```

```

section Fiber $ColSecTag2 {
# Define the core patch
patch quad $concCore2 $nfcoreZC $nfcoreYC -$scoreYC $scoreZC -$scoreYC -$scoreZC
$scoreYC -$scoreZC $scoreYC $scoreZC
# Define the four cover patches
patch quad $concCover2 2 $nfcoverYC -$coverYC $coverZC -$scoreYC $scoreZC $scoreYC
$scoreZC $coverYC $coverZC
patch quad $concCover2 2 $nfcoverYC -$scoreYC -$scoreZC -$coverYC -$coverZC
$coverYC -$coverZC $scoreYC -$scoreZC
patch quad $concCover2 $nfcoverZC 2 -$coverYC $coverZC -$coverYC -$coverZC -
$scoreYC -$scoreZC -$scoreYC $scoreZC
patch quad $concCover2 $nfcoverZC 2 $scoreYC $scoreZC $scoreYC -$scoreZC $coverYC -
$coverZC $coverYC $coverZC
# define reinforcing layers
layer straight $steelRebar2 $nBarsC $Area1S $scoreYC $scoreZC $scoreYC -$scoreZC;
layer straight $steelRebar2 $nBarsC $Area1S 0.0 $scoreZC 0.0 -$scoreZC;

```

```

layer straight $steelRebar2 $nBarsC $Area1S -$scoreYC $scoreZC -$scoreYC -
$scoreZC;
}

section Fiber $ColSecTag345 {
# Define the core patch
patch quad $concCore345 $nfcoreZC $nfcoreYC -$scoreYC $scoreZC -$scoreYC -$scoreZC
$scoreYC -$scoreZC $scoreYC $scoreZC
# Define the four cover patches
patch quad $concCover345 2 $nfcoverYC -$coverYC $coverZC -$scoreYC $scoreZC
$scoreYC $scoreZC $coverYC $coverZC
patch quad $concCover345 2 $nfcoverYC -$scoreYC -$scoreZC -$coverYC -$coverZC
$coverYC -$coverZC $scoreYC -$scoreZC
patch quad $concCover345 $nfcoverZC 2 -$coverYC $coverZC -$coverYC -$coverZC -
$scoreYC -$scoreZC -$scoreYC $scoreZC
patch quad $concCover345 $nfcoverZC 2 $scoreYC $scoreZC $scoreYC -$scoreZC $coverYC
-$coverZC $coverYC $coverZC
# define reinforcing layers
layer straight $steelRebar345 $nBarsC $Area1S $scoreYC $scoreZC $scoreYC -$scoreZC;
layer straight $steelRebar345 $nBarsC $Area1S 0.0 $scoreZC 0.0 -$scoreZC;

layer straight $steelRebar345 $nBarsC $Area1S -$scoreYC $scoreZC -$scoreYC -
$scoreZC;
}

# Beams section
# fiber section parameters
set HsecB 0.55;
set BsecB 0.35;
set coverB 0.03;
set nfB 10;
set nBarsB [expr ($nfB/2)-1];
set nBarsMidB [expr $nfB-2*$nBarsB];
set fiB 0.024;
set barArea1B [expr (3.14159*pow($fiB,2)/4)];
set barAreaB [expr $nfB*(3.14159*pow($fiB,2)/4)];
set barAreaTopB [expr ($barAreaB/2.0)-$barArea1B];
set barAreaBotB [expr ($barAreaB/2.0)-$barArea1B];

```

```

set barAreaMidB [expr ($barArea1B*$nBarsMidB)]
set coverYB [expr $HsecB/2.0];
set coverZB [expr $BsecB/2.0];
set coreYB [expr $coverYB-$coverB];
set coreZB [expr $coverZB-$coverB];
set nfcoverYB 16;
set nfcoverZB 4;
set nfcoreYB 16;
set nfcoreZB 4;

section Fiber $BeamSecTag1 {
# Define the core patch
patch quad $concCore1 $nfcoreZB $nfcoreYB -$coreYB $coreZB -$coreYB -$coreZB
$coreYB -$coreZB $coreYB $coreZB
# Define the four cover patches
patch quad $concCover1 2 $nfcoverYB -$coverYB $coverZB -$coreYB $coreZB $coreYB
$coreZB $coverYB $coverZB
patch quad $concCover1 2 $nfcoverYB -$coreYB -$coreZB -$coverYB -$coverZB
$coreYB -$coverZB $coreYB -$coreZB
patch quad $concCover1 $nfcoverZB 2 -$coverYB $coverZB -$coverYB -$coverZB -
$coreYB -$coreZB -$coreYB $coreZB
patch quad $concCover1 $nfcoverZB 2 $coreYB $coreZB $coreYB -$coreZB $coverYB -
$coreZB $coverYB $coverZB
# define reinforcing layers
layer straight $steelRebar1 $nBarsB $barAreaTopB $coreYB $coreZB $coreYB -
$coreZB;
layer straight $steelRebar1 $nBarsMidB $barAreaMidB 0.0 $coreZB 0.0 -$coreZB;

layer straight $steelRebar1 $nBarsB $barAreaBotB -$coreYB $coreZB -$coreYB -
$coreZB;
}

section Fiber $BeamSecTag2 {
# Define the core patch
patch quad $concCore2 $nfcoreZB $nfcoreYB -$coreYB $coreZB -$coreYB -$coreZB
$coreYB -$coreZB $coreYB $coreZB
# Define the four cover patches

```

```

patch quad $concCover2 2 $nfcoverYB -$coverYB $coverZB -$coreYB $coreZB $coreYB
$coreZB $coverYB $coverZB

patch quad $concCover2 2 $nfcoverYB -$coreYB -$coreZB -$coverYB -$coverZB
$coverYB -$coverZB $coreYB -$coreZB

patch quad $concCover2 $nfcoverZB 2 -$coverYB $coverZB -$coverYB -$coverZB -
$coreYB -$coreZB -$coreYB $coreZB

patch quad $concCover2 $nfcoverZB 2 $coreYB $coreZB $coreYB -$coreZB $coverYB -
$coverZB $coverYB $coverZB

# define reinforcing layers

layer straight $steelRebar2 $nBarsB $barAreaTopB $coreYB $coreZB $coreYB -
$coreZB;

layer straight $steelRebar2 $nBarsMidB $barAreaMidB 0.0 $coreZB 0.0 -$coreZB;

layer straight $steelRebar2 $nBarsB $barAreaBotB -$coreYB $coreZB -$coreYB -
$coreZB;

}

section Fiber $BeamSecTag345 {
# Define the core patch

patch quad $concCore345 $nfcoreZB $nfcoreYB -$coreYB $coreZB -$coreYB -$coreZB
$coreYB -$coreZB $coreYB $coreZB

# Define the four cover patches

patch quad $concCover345 2 $nfcoverYB -$coverYB $coverZB -$coreYB $coreZB
$coreYB $coreZB $coverYB $coverZB

patch quad $concCover345 2 $nfcoverYB -$coreYB -$coreZB -$coverYB -$coverZB
$coverYB -$coverZB $coreYB -$coreZB

patch quad $concCover345 $nfcoverZB 2 -$coverYB $coverZB -$coverYB -$coverZB -
$coreYB -$coreZB -$coreYB $coreZB

patch quad $concCover345 $nfcoverZB 2 $coreYB $coreZB $coreYB -$coreZB $coverYB
-$coverZB $coverYB $coverZB

# define reinforcing layers

layer straight $steelRebar345 $nBarsB $barAreaTopB $coreYB $coreZB $coreYB -
$coreZB;

layer straight $steelRebar345 $nBarsMidB $barAreaMidB 0.0 $coreZB 0.0 -$coreZB;

layer straight $steelRebar345 $nBarsB $barAreaBotB -$coreYB $coreZB -$coreYB
-$coreZB;

}

# Girders section

```

```

# fiber section parameters
set HsecG 0.55;
set BsecG 0.35;
set coverG 0.03;
set nfG 10;
set nBarsG [expr ($nfG/2)-1];
set nBarsMidG [expr $nfG-2*$nBarsG];
set fiG 0.024;
set barArea1G [expr (3.14159*pow($fiG,2)/4)];
set barAreaG [expr $nfG*(3.14159*pow($fiG,2)/4)];
set barAreaTopG [expr ($barAreaG/2.0)-$barArea1G];
set barAreaBotG [expr ($barAreaG/2.0)-$barArea1G];
set barAreaMidG [expr ($barArea1G*$nBarsMidG)]
set coverYG [expr $HsecG/2.0];
set coverZG [expr $BsecG/2.0];
set coreYG [expr $coverYG-$coverG];
set coreZG [expr $coverZG-$coverG];
set nfcoverYG 16;
set nfcoverZG 4;
set nfcoreYG 16;
set nfcoreZG 4;

section Fiber $GirdSecTag1 {
# Define the core patch
patch quad $concCore1 $nfcoreZG $nfcoreYG -$coreYG $coreZG -$coreYG -$coreZG
$coreYG -$coreZG $coreYG $coreZG
# Define the four cover patches
patch quad $concCover1 2 $nfcoverYG -$coverYG $coverZG -$coreYG $coreZG $coreYG
$coreZG $coverYG $coverZG
patch quad $concCover1 2 $nfcoverYG -$coreYG -$coreZG -$coverYG -$coverZG
$coreYG -$coverZG $coreYG -$coreZG
patch quad $concCover1 $nfcoverZG 2 -$coverYG $coverZG -$coverYG -$coverZG -
$coreYG -$coreZG -$coreYG $coreZG
patch quad $concCover1 $nfcoverZG 2 $coreYG $coreZG $coreYG -$coreZG $coverYG -
$coreZG $coverYG $coverZG
# define reinforcing layers

```

```

layer straight $steelRebar1 $nBarsG $barAreaTopG $coreYG $coreZG $coreYG -
$coreZG;
layer straight $steelRebar1 $nBarsMidG $barAreaMidG 0.0 $coreZG 0.0 -$coreZG;

layer straight $steelRebar1 $nBarsG $barAreaBotG -$coreYG $coreZG -$coreYG -
$coreZG;
}

```

```

section Fiber $GirdSecTag2 {
# Define the core patch
patch quad $concCore2 $nfcoreZG $nfcoreYG -$coreYG $coreZG -$coreYG -$coreZG
$coreYG -$coreZG $coreYG $coreZG
# Define the four cover patches
patch quad $concCover2 2 $nfcoverYG -$coverYG $coverZG -$coreYG $coreZG $coreYG
$coreZG $coverYG $coverZG
patch quad $concCover2 2 $nfcoverYG -$coreYG -$coreZG -$coverYG -$coverZG
$coverYG -$coverZG $coreYG -$coreZG
patch quad $concCover2 $nfcoverZG 2 -$coverYG $coverZG -$coverYG -$coverZG -
$coreYG -$coreZG -$coreYG $coreZG
patch quad $concCover2 $nfcoverZG 2 $coreYG $coreZG $coreYG -$coreZG $coverYG -
$coverZG $coverYG $coverZG
# define reinforcing layers
layer straight $steelRebar2 $nBarsG $barAreaTopG $coreYG $coreZG $coreYG -
$coreZG;
layer straight $steelRebar2 $nBarsMidG $barAreaMidG 0.0 $coreZG 0.0 -$coreZG;

layer straight $steelRebar2 $nBarsG $barAreaBotG -$coreYG $coreZG -$coreYG -
$coreZG;
}

```

```

section Fiber $GirdSecTag345 {
# Define the core patch
patch quad $concCore345 $nfcoreZG $nfcoreYG -$coreYG $coreZG -$coreYG -$coreZG
$coreYG -$coreZG $coreYG $coreZG
# Define the four cover patches
patch quad $concCover345 2 $nfcoverYG -$coverYG $coverZG -$coreYG $coreZG
$coreYG $coreZG $coverYG $coverZG
patch quad $concCover345 2 $nfcoverYG -$coreYG -$coreZG -$coverYG -$coverZG
$coverYG -$coverZG $coreYG -$coreZG

```

```

patch quad $concCover345 $nfcoverZG 2 -$coverYG $coverZG -$coverYG -$coverZG -
$coreYG -$coreZG -$coreYG $coreZG

patch quad $concCover345 $nfcoverZG 2 $coreYG $coreZG $coreYG -$coreZG $coverYG
-$coverZG $coverYG $coverZG

# define reinforcing layers

layer straight $steelRebar345 $nBarsG $barAreaTopG $coreYG $coreZG $coreYG -
$coreZG;

layer straight $steelRebar345 $nBarsMidG $barAreaMidG 0.0 $coreZG 0.0 -$coreZG;

layer straight $steelRebar345 $nBarsG $barAreaBotG -$coreYG $coreZG -$coreYG
-$coreZG;

}

set Gc 21000000000.0;
set IzC [expr (($BsecC*$HsecC*$HsecC*$HsecC)/12)];
set IyC [expr (($HsecC*$BsecC*$BsecC*$BsecC)/12)];
set JC [expr $IzC+$IyC];
set GJ [expr $JC*$Gc];

uniaxialMaterial Elastic $IdColTors1 $GJ;
section Aggregator $TColSecTag1 $IdColTors1 T -section $ColSecTag1;

uniaxialMaterial Elastic $IdColTors2 $GJ;
section Aggregator $TColSecTag2 $IdColTors2 T -section $ColSecTag2;

uniaxialMaterial Elastic $IdColTors345 $GJ;
section Aggregator $TColSecTag345 $IdColTors345 T -section $ColSecTag345;

uniaxialMaterial Elastic $FRPIdColTors1 $GJ;
section Aggregator $TFRPColSecTag1 $FRPIdColTors1 T -section $FRPColSecTag1;

uniaxialMaterial Elastic $FRPIdColTors2 $GJ;
section Aggregator $TFRPColSecTag2 $FRPIdColTors2 T -section $FRPColSecTag2;

uniaxialMaterial Elastic $FRPIdColTors345 $GJ;

```

```
section Aggregator $TFRPColSecTag345 $FRPIdColTors345 T -section
$FRPColSecTag345;
```

```
# ELASTIC section
```

```
set HsecC 0.35;
```

```
set BsecC 0.35;
```

```
set coverC 0.03;
```

```
set AC [expr $HsecC*$BsecC];
```

```
set IzC [expr ((BsecC*$HsecC*$HsecC*$HsecC)/12)];
```

```
set IyC [expr ((HsecC*$BsecC*$BsecC*$BsecC)/12)];
```

```
set JC [expr $IzC+$IyC];
```

```
set Gc 21000000000.0;
```

```
section Elastic $EColSecTag $Ec $AC $IzC $IyC $Gc $JC
```

```
# Beams section
```

```
# fiber section parameters
```

```
set HsecB 0.55;
```

```
set BsecB 0.35;
```

```
set coverB 0.03;
```

```
set AB [expr $HsecB*$BsecB];
```

```
set IzB [expr ((BsecB*$HsecB*$HsecB*$HsecB)/12)];
```

```
set IyB [expr ((HsecB*$BsecB*$BsecB*$BsecB)/12)];
```

```
set JB [expr $IzB+$IyB];
```

```
section Elastic $EBeamSecTag $Ec $AB $IzB $IyB $Gc $JB
```

```
# Girds section
```

```
# fiber section parameters
```

```

set HsecG 0.55;
set BsecG 0.35;
set coverG 0.03;
set nfG 10;

set AG [expr $HsecG*$BsecG];
set IzG [expr (($BsecG*$HsecG*$HsecG*$HsecG)/12)];
set IyG [expr (($HsecG*$BsecG*$BsecG*$BsecG)/12)];
set JG [expr $IzG+$IyG];

section Elastic $EGirdSecTag $Ec $AG $IzG $IyG $Gc $JG

# Define nodal masses
# -----
set volCol [expr $HsecC*$BsecC*$LCol];
set volBeam [expr $HsecB*$BsecB*$LBeam];
set volGird [expr $HsecG*$BsecG*$LGird];
set gammaConc 2548.54; #kg/m3
set g 9.806; #m/s2
set pesoSpecConc [expr ($gammaConc*$g)]; #N/m3
set WCol [expr $pesoSpecConc * $volCol]; # N
set WBeam [expr $pesoSpecConc * $volBeam];
set WGird [expr $pesoSpecConc * $volGird];

set MCol [expr ($WCol/$g)]; #kg
set MBeam [expr ($WBeam/$g)]; #kg
set MGird [expr ($WGird/$g)]; #kg

set vertical 150000.0; #N
set m11 [expr (4.0*$MCol+2*$MBeam+2*$MGird)];
set m22 [expr (2.0*$MCol+2.0*$MBeam+2.0*$MGird)];

set mbase [expr 0.5*$MCol];
set m1 [expr $m11/4+$vertical/$g];

```

```

set m2 [expr $m22/4+$vertical/$g];
set P1 [expr (-1*($vertical))];
set P2 [expr (-1*($vertical))];
set QdlCol [expr $WCol/$LCol];
set QdlBeam [expr $WBeam/$LBeam];
set QdlGird [expr $WGird/$LGird];

mass 10101 $mbase $mbase $mbase 0.0 0.0 0.0;
mass 10201 $mbase $mbase $mbase 0.0 0.0 0.0;
mass 10102 $mbase $mbase $mbase 0.0 0.0 0.0;
mass 10202 $mbase $mbase $mbase 0.0 0.0 0.0;

mass 20101 $m1 $m1 $m1 0.0 0.0 0.0;
mass 20201 $m1 $m1 $m1 0.0 0.0 0.0;
mass 20102 $m1 $m1 $m1 0.0 0.0 0.0;
mass 20202 $m1 $m1 $m1 0.0 0.0 0.0;

mass 30101 $m1 $m1 $m1 0.0 0.0 0.0;
mass 30201 $m1 $m1 $m1 0.0 0.0 0.0;
mass 30102 $m1 $m1 $m1 0.0 0.0 0.0;
mass 30202 $m1 $m1 $m1 0.0 0.0 0.0;

mass 40101 $m1 $m1 $m1 0.0 0.0 0.0;
mass 40201 $m1 $m1 $m1 0.0 0.0 0.0;
mass 40102 $m1 $m1 $m1 0.0 0.0 0.0;
mass 40202 $m1 $m1 $m1 0.0 0.0 0.0;

mass 50101 $m1 0.0 $m1 0.0 0.0 0.0;
mass 50201 $m1 0.0 $m1 0.0 0.0 0.0;
mass 50102 $m1 0.0 $m1 0.0 0.0 0.0;
mass 50202 $m1 0.0 $m1 0.0 0.0 0.0;

mass 60101 $m2 0.0 $m2 0.0 0.0 0.0;
mass 60201 $m2 0.0 $m2 0.0 0.0 0.0;

```

```

mass 60102 $m2 0.0 $m2 0.0 0.0 0.0;
mass 60202 $m2 0.0 $m2 0.0 0.0 0.0;

# Define multi point constraints
# -----
set RigidDiaphragm ON ;
set Xa [expr ($NBay*$LBeam)/2];
set Za [expr ($NFrame-1)*$LGird/2];
set iMasterNode ""
for {set level 2} {$level <=[expr $NStory+1]} {incr level 1} {
    set Y [expr ($level-1)*$LCol];
    set MasterNodeID [expr 9900+$level]
    node $MasterNodeID $Xa $Y $Za;
    fix $MasterNodeID 0 1 0 1 0 1
    lappend iMasterNode $MasterNodeID
    set perpDirn 2;
    for {set frame 1} {$frame <=[expr $NFrame]} {incr frame 1} {
        for {set pier 1} {$pier <= [expr $NBay+1]} {incr pier 1} {
            set nodeID [expr $level*$Dlevel+$frame*$Dframe+$pier]
            rigidDiaphragm $perpDirn $MasterNodeID $nodeID;
        }
    }
}

# Define geometric transformation(s)
# -----
set IDColTransf 1;
set IDBeamTransf 2;
set IDGirdTransf 3;

set IDColTransf345 112;
set IDBeamTransf345 212;
set IDGirdTransf345 312;

```

```

geomTransf Linear $IDColTransf 1 0 0;
geomTransf Linear $IDBeamTransf 0 -1 0;
geomTransf Linear $IDGirdTransf 0 1 0;

geomTransf Linear $IDColTransf345 1 0 0;
geomTransf Linear $IDBeamTransf345 0 -1 0;
geomTransf Linear $IDGirdTransf345 0 1 0;

```

```
# Define element(s)
```

```
# -----
```

```

set LpIC [expr $LCol *0.05];
set LpIB [expr $LBeam *0.05];
set LpIG [expr $LGird *0.05];

```

```

set el1 [expr [lindex $argv 1] == 1 ? $TFRPColSecTag1: $TColSecTag1];
set el2 [expr [lindex $argv 2] == 1 ? $TFRPColSecTag1: $TColSecTag1];
set el3 [expr [lindex $argv 3] == 1 ? $TFRPColSecTag1: $TColSecTag1];
set el4 [expr [lindex $argv 4] == 1 ? $TFRPColSecTag1: $TColSecTag1];
set el5 [expr [lindex $argv 5] == 1 ? $TFRPColSecTag1: $TColSecTag1];
set el6 [expr [lindex $argv 6] == 1 ? $TFRPColSecTag1: $TColSecTag1];
set el7 [expr [lindex $argv 7] == 1 ? $TFRPColSecTag1: $TColSecTag1];
set el8 [expr [lindex $argv 8] == 1 ? $TFRPColSecTag1: $TColSecTag1];
set el9 [expr [lindex $argv 9] == 1 ? $TFRPColSecTag1: $TColSecTag1];
set el10 [expr [lindex $argv 10] == 1 ? $TFRPColSecTag1: $TColSecTag1];
set el11 [expr [lindex $argv 11] == 1 ? $TFRPColSecTag1: $TColSecTag1];
set el12 [expr [lindex $argv 12] == 1 ? $TFRPColSecTag1: $TColSecTag1];
set el13 [expr [lindex $argv 13] == 1 ? $TFRPColSecTag1: $TColSecTag1];
set el14 [expr [lindex $argv 14] == 1 ? $TFRPColSecTag1: $TColSecTag1];
set el15 [expr [lindex $argv 15] == 1 ? $TFRPColSecTag1: $TColSecTag1];
set el16 [expr [lindex $argv 16] == 1 ? $TFRPColSecTag1: $TColSecTag1];
set el17 [expr [lindex $argv 17] == 1 ? $TFRPColSecTag1: $TColSecTag1];
set el18 [expr [lindex $argv 18] == 1 ? $TFRPColSecTag1: $TColSecTag1];
set el19 [expr [lindex $argv 19] == 1 ? $TFRPColSecTag1: $TColSecTag1];
set el20 [expr [lindex $argv 20] == 1 ? $TFRPColSecTag1: $TColSecTag1];

```

```

# 0-1 -----
element forceBeamColumn 10121 10101 20101 $IDColTransf "HingeRadau $el1 $LpIC
$el1 $LpIC $el1"
element forceBeamColumn 10221 10201 20201 $IDColTransf "HingeRadau $el2 $LpIC
$el2 $LpIC $el2"
element forceBeamColumn 10122 10102 20102 $IDColTransf "HingeRadau $el3 $LpIC
$el3 $LpIC $el3"
element forceBeamColumn 10222 10202 20202 $IDColTransf "HingeRadau $el4 $LpIC
$el4 $LpIC $el4"
# 1-2 -----
element forceBeamColumn 20121 20101 30101 $IDColTransf "HingeRadau $el5 $LpIC
$el5 $LpIC $el5"
element forceBeamColumn 20221 20201 30201 $IDColTransf "HingeRadau $el6 $LpIC
$el6 $LpIC $el6"
element forceBeamColumn 20122 20102 30102 $IDColTransf "HingeRadau $el7 $LpIC
$el7 $LpIC $el7"
element forceBeamColumn 20222 20202 30202 $IDColTransf "HingeRadau $el8 $LpIC
$el8 $LpIC $el8"
# 2-3 -----
element forceBeamColumn 30121 30101 40101 $IDColTransf "HingeRadau $el9 $LpIC
$el9 $LpIC $el9"
element forceBeamColumn 30221 30201 40201 $IDColTransf "HingeRadau $el10 $LpIC
$el10 $LpIC $el10"
element forceBeamColumn 30122 30102 40102 $IDColTransf "HingeRadau $el11 $LpIC
$el11 $LpIC $el11"
element forceBeamColumn 30222 30202 40202 $IDColTransf "HingeRadau $el12 $LpIC
$el12 $LpIC $el12"
# 3-4 -----
element forceBeamColumn 40121 40101 50101 $IDColTransf "HingeRadau $el13 $LpIC
$el13 $LpIC $el13"
element forceBeamColumn 40221 40201 50201 $IDColTransf "HingeRadau $el14 $LpIC
$el14 $LpIC $el14"
element forceBeamColumn 40122 40102 50102 $IDColTransf "HingeRadau $el15 $LpIC
$el15 $LpIC $el15"
element forceBeamColumn 40222 40202 50202 $IDColTransf "HingeRadau $el16 $LpIC
$el16 $LpIC $el16"
# 4-5 -----

```

element forceBeamColumn 50121 50101 60101 \$IDColTransf "HingeRadau \$el17 \$LpIC \$el17 \$LpIC \$el17"

element forceBeamColumn 50221 50201 60201 \$IDColTransf "HingeRadau \$el18 \$LpIC \$el18 \$LpIC \$el18"

element forceBeamColumn 50122 50102 60102 \$IDColTransf "HingeRadau \$el19 \$LpIC \$el19 \$LpIC \$el19"

element forceBeamColumn 50222 50202 60202 \$IDColTransf "HingeRadau \$el20 \$LpIC \$el20 \$LpIC \$el20"

1 -----

element forceBeamColumn 13 20101 20201 \$IDGirdTransf "HingeRadau \$GirdSecTag1 \$LpIG \$GirdSecTag1 \$LpIG \$GirdSecTag1"

element forceBeamColumn 14 20201 20202 \$IDBeamTransf "HingeRadau \$BeamSecTag1 \$LpIB \$BeamSecTag1 \$LpIB \$BeamSecTag1"

element forceBeamColumn 15 20202 20102 \$IDGirdTransf "HingeRadau \$GirdSecTag1 \$LpIG \$GirdSecTag1 \$LpIG \$GirdSecTag1"

element forceBeamColumn 16 20102 20101 \$IDBeamTransf "HingeRadau \$BeamSecTag1 \$LpIB \$BeamSecTag1 \$LpIB \$BeamSecTag1"

2 -----

element forceBeamColumn 17 30101 30201 \$IDGirdTransf "HingeRadau \$GirdSecTag2 \$LpIG \$GirdSecTag2 \$LpIG \$GirdSecTag2"

element forceBeamColumn 18 30201 30202 \$IDBeamTransf "HingeRadau \$BeamSecTag2 \$LpIB \$BeamSecTag2 \$LpIB \$BeamSecTag2"

element forceBeamColumn 19 30202 30102 \$IDGirdTransf "HingeRadau \$GirdSecTag2 \$LpIG \$GirdSecTag2 \$LpIG \$GirdSecTag2"

element forceBeamColumn 20 30102 30101 \$IDBeamTransf "HingeRadau \$BeamSecTag2 \$LpIB \$BeamSecTag2 \$LpIB \$BeamSecTag2"

3 -----

element forceBeamColumn 21 40101 40201 \$IDGirdTransf "HingeRadau \$GirdSecTag345 \$LpIG \$GirdSecTag345 \$LpIG \$GirdSecTag345"

element forceBeamColumn 22 40201 40202 \$IDBeamTransf "HingeRadau \$BeamSecTag345 \$LpIB \$BeamSecTag345 \$LpIB \$BeamSecTag345"

element forceBeamColumn 23 40202 40102 \$IDGirdTransf "HingeRadau \$GirdSecTag345 \$LpIG \$GirdSecTag345 \$LpIG \$GirdSecTag345"

element forceBeamColumn 24 40102 40101 \$IDBeamTransf "HingeRadau \$BeamSecTag345 \$LpIB \$BeamSecTag345 \$LpIB \$BeamSecTag345"

4 -----

element forceBeamColumn 333 50101 50201 \$IDGirdTransf "HingeRadau \$GirdSecTag345 \$LpIG \$GirdSecTag345 \$LpIG \$GirdSecTag345"

element forceBeamColumn 343 50201 50202 \$IDBeamTransf "HingeRadau \$BeamSecTag345 \$LpIB \$BeamSecTag345 \$LpIB \$BeamSecTag345"

```

element forceBeamColumn 353 50202 50102 $IDGirdTransf "HingeRadau
$GirdSecTag345 $LpIG $GirdSecTag345 $LpIG $GirdSecTag345"

element forceBeamColumn 363 50102 50101 $IDBeamTransf "HingeRadau
$BeamSecTag345 $LpIB $BeamSecTag345 $LpIB $BeamSecTag345"

# 5 -----

element forceBeamColumn 373 60101 60201 $IDGirdTransf "HingeRadau
$GirdSecTag345 $LpIG $GirdSecTag345 $LpIG $GirdSecTag345"

element forceBeamColumn 383 60201 60202 $IDBeamTransf "HingeRadau
$BeamSecTag345 $LpIB $BeamSecTag345 $LpIB $BeamSecTag345"

element forceBeamColumn 393 60202 60102 $IDGirdTransf "HingeRadau
$GirdSecTag345 $LpIG $GirdSecTag345 $LpIG $GirdSecTag345"

element forceBeamColumn 403 60102 60101 $IDBeamTransf "HingeRadau
$BeamSecTag345 $LpIB $BeamSecTag345 $LpIB $BeamSecTag345"

# Define damping parameters
rayleigh 0. 0. 0. 0.1759; # set damp [expr 2*DampRatio*(1/Tn)]

#puts "Analysis: Modal"
# AnalysisOptn: Modal
# -----
# Constraint Handler
constraints Transformation
# DOF Numberer
numberer Plain
# System of Equations
system UmfPack -lvalueFact 10
# Convergence Test
#test NormDispIncr +1.000000E-12 25 0 2
# Solution Algorithm
algorithm Newton
# Integrator
integrator Newmark +5.000000E-01 +2.500000E-01
# Analysis Type
analysis Transient

set numeigenvalues 3;

```

```

set lambda [eigen -fullGenLapack $numeigenvalues];
set omega {};
set f {};
set T {};
set pi 3.141593;
foreach lam $lambda {
    lappend omega [expr sqrt($lam)];
    lappend f [expr sqrt($lam)/(2*$pi)];
    lappend T [expr (2*$pi)/sqrt($lam)];
}

```

```

pattern Plain 1 Linear {

```

```

load 20101 0.0 $P1 0.0 0.0 0.0 0.0;
load 20201 0.0 $P1 0.0 0.0 0.0 0.0;
load 20102 0.0 $P1 0.0 0.0 0.0 0.0;
load 20202 0.0 $P1 0.0 0.0 0.0 0.0;

```

```

load 30101 0.0 $P1 0.0 0.0 0.0 0.0;
load 30201 0.0 $P1 0.0 0.0 0.0 0.0;
load 30102 0.0 $P1 0.0 0.0 0.0 0.0;
load 30202 0.0 $P1 0.0 0.0 0.0 0.0;

```

```

load 40101 0.0 $P1 0.0 0.0 0.0 0.0;
load 40201 0.0 $P1 0.0 0.0 0.0 0.0;
load 40102 0.0 $P1 0.0 0.0 0.0 0.0;
load 40202 0.0 $P1 0.0 0.0 0.0 0.0;

```

```

load 50101 0.0 $P1 0.0 0.0 0.0 0.0;
load 50201 0.0 $P1 0.0 0.0 0.0 0.0;
load 50102 0.0 $P1 0.0 0.0 0.0 0.0;
load 50202 0.0 $P1 0.0 0.0 0.0 0.0;

```

```

load 60101 0.0 $P2 0.0 0.0 0.0 0.0;

```

```
load 60201 0.0 $P2 0.0 0.0 0.0 0.0;
load 60102 0.0 $P2 0.0 0.0 0.0 0.0;
load 60202 0.0 $P2 0.0 0.0 0.0 0.0;
```

```
eleLoad -ele 10121 -type -beamUniform 0. 0. -$QdlCol;
eleLoad -ele 10122 -type -beamUniform 0. 0. -$QdlCol;
eleLoad -ele 10221 -type -beamUniform 0. 0. -$QdlCol;
eleLoad -ele 10222 -type -beamUniform 0. 0. -$QdlCol;
eleLoad -ele 20121 -type -beamUniform 0. 0. -$QdlCol;
eleLoad -ele 20122 -type -beamUniform 0. 0. -$QdlCol;
eleLoad -ele 20221 -type -beamUniform 0. 0. -$QdlCol;
eleLoad -ele 20222 -type -beamUniform 0. 0. -$QdlCol;
eleLoad -ele 30121 -type -beamUniform 0. 0. -$QdlCol;
eleLoad -ele 30122 -type -beamUniform 0. 0. -$QdlCol;
eleLoad -ele 30221 -type -beamUniform 0. 0. -$QdlCol;
eleLoad -ele 30222 -type -beamUniform 0. 0. -$QdlCol;
eleLoad -ele 40121 -type -beamUniform 0. 0. -$QdlCol;
eleLoad -ele 40122 -type -beamUniform 0. 0. -$QdlCol;
eleLoad -ele 40221 -type -beamUniform 0. 0. -$QdlCol;
eleLoad -ele 40222 -type -beamUniform 0. 0. -$QdlCol;
eleLoad -ele 50121 -type -beamUniform 0. 0. -$QdlCol;
eleLoad -ele 50122 -type -beamUniform 0. 0. -$QdlCol;
eleLoad -ele 50221 -type -beamUniform 0. 0. -$QdlCol;
eleLoad -ele 50222 -type -beamUniform 0. 0. -$QdlCol;
```

```
eleLoad -ele 13 -type -beamUniform -$QdlGird 0.;
eleLoad -ele 14 -type -beamUniform -$QdlBeam 0.;
eleLoad -ele 15 -type -beamUniform -$QdlGird 0.;
eleLoad -ele 16 -type -beamUniform -$QdlBeam 0.;
eleLoad -ele 17 -type -beamUniform -$QdlGird 0.;
eleLoad -ele 18 -type -beamUniform -$QdlBeam 0.;
eleLoad -ele 19 -type -beamUniform -$QdlGird 0.;
eleLoad -ele 20 -type -beamUniform -$QdlBeam 0.;
eleLoad -ele 21 -type -beamUniform -$QdlGird 0.;
```

```

eleLoad -ele 22 -type -beamUniform -$QdlBeam 0.;
eleLoad -ele 23 -type -beamUniform -$QdlGird 0.;
eleLoad -ele 24 -type -beamUniform -$QdlBeam 0.;
eleLoad -ele 333 -type -beamUniform -$QdlGird 0.;
eleLoad -ele 343 -type -beamUniform -$QdlBeam 0.;
eleLoad -ele 353 -type -beamUniform -$QdlGird 0.;
eleLoad -ele 363 -type -beamUniform -$QdlBeam 0.;
eleLoad -ele 373 -type -beamUniform -$QdlGird 0.;
eleLoad -ele 383 -type -beamUniform -$QdlBeam 0.;
eleLoad -ele 393 -type -beamUniform -$QdlGird 0.;
eleLoad -ele 403 -type -beamUniform -$QdlBeam 0.;
}
set TestType EnergyIncr;
set Tol 1.e-03;
set maxNumIter 150;
test $TestType $Tol $maxNumIter;
constraints Transformation;
numberer Plain;
system BandGeneral;
algorithm KrylovNewton;
set NstepGravity 10;
set DGravity [expr 1./$NstepGravity];
integrator LoadControl $DGravity;
analysis Static;
set Gravity [analyze $NstepGravity];
if {$Gravity != 0} {
    puts "Gravity loads not OK"
}
loadConst -time 0.0

# Set up the acceleration records for Norcia fault normal and fault parallel
set ScaleFactor 9.81;
set NorciaEWrr "Path -filePath Norcia_EW.txt -dt 0.005 -factor $ScaleFactor";

```

```

set NorciaNSrr "Path -filePath Norcia_NS.txt -dt 0.005 -factor $ScaleFactor";
# Define the excitation using the Norcia ground motion records
pattern UniformExcitation 222 1 -accel $NorciaNSrr
pattern UniformExcitation 333 3 -accel $NorciaNSrr

# Define Recorder(s)
# -----
recorder Node -file 1_node8(OpenSees).txt -time -node 20202 -dof 1 disp;
recorder Node -file 1_node12(OpenSees).txt -time -node 30202 -dof 1 disp;
recorder Node -file 1_node16(OpenSees).txt -time -node 40202 -dof 1 disp;
recorder Node -file 1_node20(OpenSees).txt -time -node 50202 -dof 1 disp;
recorder Node -file 1_node24(OpenSees).txt -time -node 60202 -dof 1 disp;

recorder Node -file 1_node8(OpenSees)Z.txt -time -node 20202 -dof 3 disp;
recorder Node -file 1_node12(OpenSees)Z.txt -time -node 30202 -dof 3 disp;
recorder Node -file 1_node16(OpenSees)Z.txt -time -node 40202 -dof 3 disp;
recorder Node -file 1_node20(OpenSees)Z.txt -time -node 50202 -dof 3 disp;
recorder Node -file 1_node24(OpenSees)Z.txt -time -node 60202 -dof 3 disp;

recorder Drift -file [lindex $argv 0]/0.txt -time -iNode 10202 -jNode 20202 -
dof 1 -perpDirn 2;
recorder Drift -file [lindex $argv 0]/1.txt -time -iNode 20202 -jNode 30202 -
dof 1 -perpDirn 2;
recorder Drift -file [lindex $argv 0]/2.txt -time -iNode 30202 -jNode 40202 -
dof 1 -perpDirn 2;
recorder Drift -file [lindex $argv 0]/3.txt -time -iNode 40202 -jNode 50202 -
dof 1 -perpDirn 2;
recorder Drift -file [lindex $argv 0]/4.txt -time -iNode 50202 -jNode 60202 -
dof 1 -perpDirn 2;

algorithm KrylovNewton
system SparseGeneral -piv
constraints Transformation;
integrator Newmark 0.7 0.3
numberer RCM
analysis Transient

```

```

set Tol 1.0e-8;
set maxNumIter 100;
set printFlag 0;
set NewmarkGamma 0.5;
set NewmarkBeta 0.25;
set TestType EnergyIncr;

set NstepsCheck 4000
set Nsteps 4000
set dt 0.005
set algorithmType NewtonLineSearch;
for {set ik 1} {$ik <= 3} {incr ik 1} {
#puts "$ik"
set ok [analyze $Nsteps $dt]
# Convergence
if {$ok != 0} {
#puts "Trying Bisection ...";
algorithm NewtonLineSearch <-type Bisection>;
set ok [analyze $Nsteps $dt]
algorithm $algorithmType;
};
if {$ok != 0} {
#puts "Trying Secant ...";
algorithm NewtonLineSearch <-type Secant>;
set ok [analyze $Nsteps $dt]
algorithm $algorithmType;
};
if {$ok != 0} {
#puts "Trying RegulaFalsi ...";
algorithm NewtonLineSearch <-type RegulaFalsi>;
set ok [analyze $Nsteps $dt]
algorithm $algorithmType;
};
};

```

```

if {$ok != 0} {
#puts "Trying KrylovNewton ...";
algorithm KrylovNewton;
set ok [analyze $Nsteps $dt]
algorithm $algorithmType;
};
if {$ok != 0} {
#puts "Trying Newton ...";
algorithm Newton;
set ok [analyze $Nsteps $dt]
algorithm $algorithmType;
};
if {$ok != 0} {
#puts "Trying BFGS ...";
algorithm BFGS;
set ok [analyze $Nsteps $dt]
algorithm $algorithmType;
};
if {$ok != 0} {
#puts "Trying Broyden ...";
algorithm Broyden;
set ok [analyze $Nsteps $dt]
algorithm $algorithmType;
};
if {$ok != 0} {
#puts "Trying HHT 0.9 ...";
integrator HHT 0.9;
set ok [analyze $Nsteps $dt]
# integrator TRBDF2;
integrator Newmark $NewmarkGamma $NewmarkBeta;
};
if {$ok != 0} {
#puts "Trying OS ...";
integrator AlphaOS 1.00;

```

```

algorithm Linear;
set ok [analyze $Nsteps $dt]
# integrator TRBDF2;
integrator Newmark $NewmarkGamma $NewmarkBeta;
algorithm $algorithmType;
};
if {$ok != 0} {
#puts "Trying OSG ...";
integrator AlphaOSGeneralized 1.00;
algorithm Linear;
set ok [analyze $Nsteps $dt]
# integrator TRBDF2;
integrator Newmark $NewmarkGamma $NewmarkBeta;
algorithm $algorithmType;
};
if {$ok != 0} {
#puts "Trying more iterations...";
test $TestType $Tol 1000 $printFlag;
set ok [analyze $Nsteps $dt]
test $TestType $Tol $maxNumIter $printFlag;
};
if {$ok != 0} {
#puts "Trying tolerance 1.0e-7 ...";
test $TestType 1.0e-7 $maxNumIter 0;
set ok [analyze $Nsteps $dt]
test $TestType $Tol $maxNumIter $printFlag;
};
if {$ok != 0} {
#puts "Trying tolerance 1.0e-6 ...";
test $TestType 1.0e-6 $maxNumIter 0;
set ok [analyze $Nsteps $dt]
test $TestType $Tol $maxNumIter $printFlag;
};
if {$ok != 0} {

```

```

#puts "Trying tolerance 1.0e-5 ...";
test $TestType 1.0e-5 $maxNumIter 0;
set ok [analyze $Nsteps $dt]
test $TestType $Tol $maxNumIter $printFlag;
};
if {$ok != 0} {
#puts "Trying tolerance 1.0e-3 ...";
test $TestType 1.0e-3 $maxNumIter 0;
set ok [analyze $Nsteps $dt]
test $TestType $Tol $maxNumIter $printFlag;
};
if {$ok != 0} {
#puts "Trying tolerance 1.0e-3 ...";
test $TestType 1.0e-3 $maxNumIter 0;
set ok [analyze $Nsteps [expr $dt/2.]]
test $TestType $Tol $maxNumIter $printFlag;
};
if {$ok != 0} {
#puts "Trying tolerance 1.0e-3 ...";
test $TestType 1.0e-3 $maxNumIter 0;
set ok [analyze $Nsteps [expr $dt/4.]]
test $TestType $Tol $maxNumIter $printFlag;
};
if {$ok != 0} {
#puts "Trying tolerance 1.0e-3 ...";
test $TestType 1.0e-3 $maxNumIter 0;
set ok [analyze $Nsteps [expr $dt/8.]]
test $TestType $Tol $maxNumIter $printFlag;
};
if {$ok != 0} {
#puts "Trying tolerance 1.0e-3 ...";
test $TestType 1.0e-3 $maxNumIter 0;
set ok [analyze $Nsteps [expr $dt/16.]]
test $TestType $Tol $maxNumIter $printFlag;
};

```

```
};  
if {$ok != 0} {  
#puts "Trying tolerance 1.0e-3 ...";  
test $TestType 1.0e-3 $maxNumIter 0;  
set ok [analyze $Nsteps [expr $dt/32.]]  
test $TestType $Tol $maxNumIter $printFlag;  
};  
if {$ok != 0} {  
#puts "Trying tolerance 1.0e-3 ...";  
test $TestType 1.0e-3 $maxNumIter 0;  
set ok [analyze $Nsteps [expr $dt/64.]]  
test $TestType $Tol $maxNumIter $printFlag;  
};  
  
}
```

9. BIBLIOGRAPHY

ACI-Committee-440, A. C. I.-. (2008). "Guide for the Design and Construction of Externally Bonded FRP Systems for Strengthening Concrete Structures."

Braga, F., et al. (2006). "Analytical stress–strain relationship for concrete confined by steel stirrups and/or FRP jackets." Journal of Structural Engineering **132**(9): 1402-1416.

Cimellaro, G. P. and S. Marasco (2015). "A computer-based environment for processing and selection of seismic ground motion records: OPENSIGNAL." Frontiers in Built Environment **1**: 17.

Cimellaro, G. P. and S. Marasco (2016). Fundamentals of Dynamic of Structures and Earthquake Engineering, Universitas studiorum.

CNR, C. N. d. R. (2014). "Istruzioni per la Progettazione, l'Esecuzione ed il Controllo di Interventi di Consolidamento Statico mediante l'utilizzo di Compositi Fibrorinforzati."

CSI, S. V. (2002). "8. Integrated finite element analysis and design of structures basic analysis reference manual." Computers and Structures Inc, Berkeley (CA, USA) **9**.

Ferrari, P. (2018). "Convergence of Genetic Algorithms - A Markov chain analysis of the limit behavior of the Canonical Genetic Algorithm."

fib (2001). "Externally bonded FRP reinforcement for RC structures."

Humar, J. (2012). Dynamics of structures, CRC press.

Karsan, I. D. and J. O. Jirsa (1969). "Behavior of concrete under compressive loadings." Journal of the Structural Division.

Lam, L. and J. Teng (2003). "Design-oriented stress–strain model for FRP-confined concrete." Construction and building materials **17**(6-7): 471-489.

McKenna, F. (2011). "OpenSees: a framework for earthquake engineering simulation." Computing in Science & Engineering **13**(4): 58-66.

NTC (2018). "Norme tecniche per le costruzioni."

Paulay, T. and M. N. Priestley (1992). "Seismic design of reinforced concrete and masonry buildings."

Scott, M. H. (2011). "Numerical integration options for the force-based beam-column element in OpenSees." Force-Based Element Integration Options in OpenSees: 1-7.

Scott, M. H. and G. L. Fenves (2006). "Plastic hinge integration methods for force-based beam-column elements." Journal of Structural Engineering **132**(2): 244-252.

Spacone, E., et al. (1996). "Fibre beam-column model for non-linear analysis of R/C frames: Part I. Formulation." Earthquake Engineering & Structural Dynamics **25**(7): 711-725.

Terzic, V. (2011). "Force-based Element vs. Displacementbased Element." University of California, Berkeley, USA, December.

UNI-EN (1998). Progettazione delle strutture per la resistenza sismica. Valutazione e adeguamento degli edifici.

Vanderplaats, G. N. (2001). Numerical optimization techniques for engineering design, Vanderplaats Research and Development, Incorporated.

Zou, X., et al. (2007). "Optimal performance-based design of FRP jackets for seismic retrofit of reinforced concrete frames." Composites Part B: Engineering **38**(5-6): 584-597.

