FINITE ELEMENT AND LIMIT ANALYSIS
OF THE LARGE SCALE MODEL OF MUSTAFA PASHA MOSQUE IN SKOPJE
STRENGTHENED WITH FRP

R. Landolfo, F. Portioli, O. Mammana and F.M. Mazzolani

University of Naples “Federico II”, Italy

ABSTRACT
This paper is concerned with the analysis of the seismic performance of the historical monument Mustafa Pasha Mosque in Skopje strengthened with FRP. The study was carried out by means of finite element and limit analysis. The main aim of numerical investigations was to control the response of the structure during the shaking table tests carried out on both the original and reinforced large scale model of the Mosque that was built at the IZIIS Laboratory in Skopje. In order to properly evaluate the structural interactions among the different parts of the specimen, a three dimensional finite element model of the Mosque was implemented. All the main elements of the building were simulated accurately, including the openings and the pendentives connecting the walls with the dome. In addition, a simplified modelling approach based on the limit equilibrium analysis of collapse mechanisms was developed in order to evaluate the ultimate load of both original and reinforced model. The results obtained from finite element and limit analysis were compared with the experimental outcome, showing the differences between the predicted and measured seismic capacity for each phase of the test programme. The evolution of partial and global collapse mechanisms observed during the tests has been also analyzed on the basis of numerical results.

KEYWORDS
FRP, masonry, strengthening, FE analysis, limit analysis.

INTRODUCTION
The present work is a part of the research project “Earthquake protection of historical buildings by reversible mixed technologies”, whose acronym is PROHITECH, which falls into Sixth Framework Programme of European Commission. The objective of the study was to evaluate the effectiveness of FRP in seismic strengthening of the Mustafa Pasha Mosque in Skopje.

Figure 1. The large scale model of the original and strengthened Mosque built at the IZIIS Laboratory in Skopje.
To investigate the seismic behaviour of the Mosque, shaking table tests were carried out on a large scale model which was built at the IZIIS Laboratory (Gramatikov et al. 2007). The scale factor of the model was 1:6 and the masonry was made of limestone and clay bricks assembled with lime mortar, using the same materials and construction principles of the full scale prototype (Figures 1, 2). As a consequence, the adopted modelling approach neglected gravity forces, which is a proper assumption when stresses induced by self weight are negligible with respect to stresses induced by seismic forces.

The aim of the experimental investigation was to characterize the dynamic behaviour of the structure with and without reinforcement and to evaluate the effects of retrofitting on both the Minaret and Mosque. The strengthening of the different parts of the building using FRPs was carried out in three different steps and the experimental investigation was organized in corresponding phases: the testing of the original model (phase I); the testing of the model with strengthened Minaret (phase II); the testing of the strengthened Mosque (phase III). The Minaret was strengthened with four strips of CFRP, placed along vertical direction, and with five confining wraps. The width of longitudinal strips was 150mm and the width of confining ones was 100mm.

The Mosque was reinforced by means of three wraps placed at the base of the dome, around the drum and at the top of the shear walls, with widths equal to 500mm and 100mm respectively. A couple of carbon rods was also inserted in two longitudinal mortar joints around the shear walls at different levels, namely on the top and under the second row of openings. Epoxy resin was used to apply FRP strengthening. The shaking table tests were carried out by using the displacement time history corresponding to the N-S component of the Petrovac record of 1979 Montenegro earthquake. The earthquake excitation was applied in the direction of x-axis. To follow the dynamic response during the seismic shaking table testing, the model was instrumented at characteristic points with accelerometers and transducers for measuring absolute displacements as well as relative diagonal deformation of the walls in direction of the excitation (Krstevska et al. 2007).

In order to control the response of the large scale model during the shaking table tests performed at the IZIIS Laboratory in Skopje, different modelling approaches were used. In particular, a finite element model was implemented and different collapse mechanisms were analyzed according to kinematic approach of limit analysis.

The main aim of numerical investigations was to evaluate the peak ground accelerations to be assigned to the shaking table during the test phases of experimental programme that were devoted to the assessment of the seismic strength of both the original and strengthened structure. In particular, it was important to control the damage on the original Minaret and on the Mosque in order to prevent collapse before the application of FRP strengthening. Moreover, the numerical models were used to determine suitable FRP strengthening schemes based on the calculated tensile stress distributions.

To assess the seismic capacity of the structure and the effectiveness of FRP reinforcement, nonlinear pushover analyses were carried out by means of the implemented finite element model. The numerical simulations were also used to support the analysis of collapse mechanisms that were observed on the prototype.
The results obtained from finite element analysis were used to develop and validate a simplified modelling procedure based on the study of collapse mechanisms.

In the following, a detailed description of the implemented models and of the corresponding obtained results is reported.

**FE ANALYSIS**

**Types of analysis, geometric modelling and meshing**

The finite element analysis of the large scale model was carried out using the ANSYS software. Push-over analyses were performed for the evaluation of structural capacity, both for original and reinforced model. The numerical model was generated by importing in the FE program a three dimensional solid model of the Mosque that was created in a computer aided design system.

In order to properly evaluate the structural interactions among the different parts, the implemented geometrical model reproduces all the main elements of the building accurately, including the openings and the pendentives connecting the walls with the dome.

Two separate FE models were implemented for the Minaret and the Mosque, respectively (Figure 3). In the last case, the symmetry of the model along the vertical plane parallel to the direction of the input displacement was considered, in order to save CPU time for solving non-linear equations. To place the reinforcement, the outer surfaces of the Mosque and Minaret were divided into different areas corresponding both to the FRP sheets and bars. The properties of overlapping areas were set up in order to match the meshes corresponding to masonry and FRP reinforcement elements.

![Figure 3. The FE model of the reinforced Mosque and Minaret implemented for non-linear pushover analysis.](image)

The whole masonry structure was discretized with tetrahedral 3D solid elements considering a mesh size of 100.0 mm in the case of Mosque model. SOLID45 elements were used for masonry and SHELL181 elements were employed to model the FRP sheets. The shells corresponding to the FRP reinforcement were directly overlapped to the masonry bricks and no interface elements were considered. The parts of the Mosque model reinforced with bars were modelled with elastic solid elements through the whole thickness of the shear walls with Young’s modulus equal to that of FRP material.

**Material modelling**

The material properties adopted for the two finite element models are reported in Table 1. In particular, the elastic parameters are referred to the values that have been calibrated on the basis of first random vibration tests performed on the original undamaged structure. The strength properties were determined on the basis of both compression and shear experimental tests carried out on masonry wall samples.

As far as structural masonry, the elastic-perfectly plastic Drucker-Prager material model was considered. In the case of Mosque, cohesion $c$ and angle of internal friction $\phi$ were calibrated in order to obtain values of 0.05 MPa and 1.0 MPa for tensile ($f_t$) and compressive strengths ($f_c$). For Minaret, the considered values for $f_t$ and $f_c$ were 0.1 and 1.0 MPa. In order to assign an associative flow rule for plastic strains, the assumed dilatancy angle $\delta$ was equal to $\phi$. With regard to composites, an elastic material model was considered. In particular, the adopted...
Young’s modulus for sheets is equal to 240 GPa and the considered equivalent thickness is 1.0 mm, according to the nominal mechanical properties provided by the manufacturer.

Table 1. Masonry material properties assumed in numerical simulations of Minaret and Mosque.

<table>
<thead>
<tr>
<th></th>
<th>Minaret</th>
<th>Mosque</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>γ (kN/m³)</td>
<td>E (MPa)</td>
</tr>
<tr>
<td></td>
<td>19.0</td>
<td>16000</td>
</tr>
</tbody>
</table>

Load modelling and boundary conditions

With regard to the load modelling approach implemented in pushover analyses for the evaluation of seismic capacity, a uniform and a linear acceleration distribution along the horizontal direction was applied to the FE models of Mosque and Minaret, respectively. As far as the boundary conditions are concerned, full restraints were assumed at the base of the structure in the performed analyses.

Results of the non-linear pushover numerical analyses

The assessment of the ultimate seismic strength of both the original and reinforced large scale model was carried out by means of nonlinear pushover analyses performed on the implemented FE model.

On the basis of obtained numerical results, the evolution of damage distribution till collapse load on the investigated prototype has to be ascribed to the attainment of tensile or shear strength, while the compressive resistance is never exceeded. The collapse loads corresponding to the original and reinforced prototype were determined by checking the attainment of the maximum plastic strain calculated for masonry on the basis of the FE models calibrated against shear tests. With regard to the Minaret, the results of numerical analysis show that the reinforcement increases the ultimate strength in terms of top acceleration from the value of 0.31g corresponding to the original model to 1.2g and the collapse mechanism shifts to the upper part without strengthening. As far as the Mosque is concerned and according to the implemented numerical model, the original prototype collapses with a mixed pier/spandrel mechanism of the vertical bearing structures, that is typical of weakly coupled perforated walls (Figure 4a). On the basis of the numerical results, the evolution of collapse mechanism can be divided in different phases. According to the numerical model, the first diagonal tension cracks occur in the shear walls, namely in the spandrels between the first and second row of openings from the basement. In this phase, the damage also develops at the base of the walls perpendicular to the direction of ground motion, owing to bending stresses induced by out plane horizontal loads. In the second step, the damage extends to the upper spandrels, between the second and third row of opening in the shear walls parallel to the direction of seismic loads. In particular, it develops up to the dome, among the openings in the supporting polygonal drum and the ones in the shear walls. Finally, the seismic strength of the structure is attained when the central wall at the base of the Mosque and the lateral piers collapse for shear and bending mechanisms, respectively.

The study of collapse mechanism, obtained from numerical analysis on both original and reinforced Mosque, allowed the effectiveness of FRP reinforcement to be analyzed.

Figure 4. Distribution of first principal plastic strains on the original (a) and reinforced (b) large scale model of the Mosque at collapse load.
The wraps around the dome and the top of shear walls fully prevent the propagation of cracks from the bottom part to the drum, as shown in Figure 4b. The role played by FRP bars in the shear walls is to stiffen and strengthen the spandrels in order to form a sort of reinforced masonry beams at different levels able to distribute the seismic action among the piers. The numerical model of the reinforced Mosque shows that the collapse mechanism turns from a mixed into a weak piers/strong spandrel type.

LIMIT ANALYSIS

The results of FE simulations were used to develop and validate a simplified calculation procedure based on the analysis of collapse mechanisms in accordance to limit analysis assumptions. In particular, the considered collapse mechanisms were selected on the basis of finite element results (Figure 5). The collapse loads were calculated by considering a uniform and a linear distribution for horizontal accelerations in the case of Mosque and Minaret, respectively. To take into account the effects of cohesion along the separation surfaces of the considered collapse mechanisms, a rigid perfectly plastic behaviour in tension with unlimited compressive strength was considered for joints. The values assumed for tensile strength $f_t$ were 0.05 and 0.1 for Mosque and Minaret models, in accordance with finite element analysis.

\[ \sum F_j \cdot \delta u_j = \sum W_i \cdot \delta u_{W_i} + \alpha \cdot \sum W_i \cdot \delta u_{aW_i} \]

(1)

where $F_j$ is the internal cohesion force on separation surface $j$ of collapse mechanism, $W_i$ is the gravity load on body $i$, $\alpha$ is the multiplier of horizontal seismic action $aW_i$ and $\delta u$ is the virtual displacement rate along load direction. To evaluate the resisting loads $W_i$ and the position of gravity centres for the calculation of virtual displacements $\delta u$ of the rigid bodies involved in the selected collapse mechanisms, a three dimensional model of the Mosque was generated in AutoCad. The ultimate loads obtained from analysis of collapse mechanisms are reported in Table 2.

COMPARISON WITH EXPERIMENTAL TEST

The results of FE model and limit analysis were compared with experimental tests. The distribution of first principal plastic strains predicted by the numerical model fits well the crack patterns observed on the large scale model of the Mosque during shaking table tests. As shown in Figure 6, in the case of the original Mosque cracks formed on the spandrels between the openings up to the tambour, while in the reinforced model cracks on the bearing walls were observed at the base of the structure, according to the predicted damage pattern. The comparison between maximum top accelerations is shown in Table 2. Also in this case, the numerical and experimental results are in good agreement. Generally, the numerical results overestimate the experimental strength in the case of original large scale model. With this regard, it should be noted that in order to use the same specimen for the different phases of testing programme the experimental tests on the original model were stopped just before the overall collapse mechanism was formed.
Figure 6. Crack patterns observed during shaking table tests on the Mosque without FRP (a) and with FRP (b).

Table 2. Predicted top accelerations and corresponding experimental values for original and reinforced large scale model.

<table>
<thead>
<tr>
<th>MINARET</th>
<th>MOSQUE</th>
</tr>
</thead>
<tbody>
<tr>
<td>w/o FRP</td>
<td>with FRP</td>
</tr>
<tr>
<td>Top acceleration</td>
<td>Experimental</td>
</tr>
<tr>
<td></td>
<td>FE model</td>
</tr>
<tr>
<td></td>
<td>Limit analysis</td>
</tr>
</tbody>
</table>

CONCLUSIONS

In the present study, a numerical investigation on the seismic strength of the masonry large scale model of the Mustafa Pasha Mosque reinforced with FRP has been presented. In order to support the experimental test set-up, to design the FRP reinforcements and to analyze their effects on the prototype, different types of numerical analysis and modelling approaches were implemented, including finite element and limit analysis.

In particular, the implemented finite element model allowed the evolution of partial and global collapse mechanisms observed during the tests to be analyzed. The FE model was also used to develop a simplified calculation procedure based on the analysis of collapse mechanisms. The results of numerical investigation have been compared with the experimental outcome. Generally, a good agreement between the behaviour predicted by numerical models and test results was observed, both in terms of ultimate seismic capacity and collapse mechanisms. Further studies are in progress, mainly devoted to refine the calibration of FE model by implementing non linear time history analysis with other types of material models for masonry, such as smeared crack. The calibrated non-linear numerical models will represent a reliable tool both for the design optimization of FRPs and for the evaluation of the effectiveness of other types of strengthening on the original Mosque.

ACKNOWLEDGMENTS

It is significantly acknowledged the financial support of the European Commission (grant No. INCO-CT-2002-509119), for funding the research project PROHITECH (Earthquake PROtection of HIstorical Buildings by Reversible Mixed TECHnologies), which is the main framework of the experimental and numerical activity presented in this paper.

REFERENCES
