CHAPTER 6

ANALYTICAL INVESTIGATION

6.1 INTRODUCTION

The finite element method (FEM) is a numerical discretization technique in structural mechanics. The basic concept for the physical interpretation of the FEM is the subdivision of the mathematical model into disjoint (non-overlapping) components of simple geometry called finite elements or elements for short. The response of each element is expressed in terms of a finite number of degrees of freedom characterized as the value of an unknown function, or functions, at a set of nodal points. The response of the mathematical model is then considered to be approximated by that of the discrete model obtained by connecting or assembling the collection of all elements. The disconnection-assembly concept occurs naturally when examining many artificial and natural systems. For example, it is easy to visualize a bridge, building, skeleton, or an airplane, engine, etc., as fabricated from simpler components. Unlike finite difference models, finite elements do not overlap in space. Some of the commercially available finite element software packages are ABAQUS, DIANA, ANSYS, ASKA, NISA, SAP, SESAMAT, GT-STRUDL, and PAFEC. ABAQUS is used for analysis in this thesis.
6.2 FEA MODELLING

The masonry model was 1 m × 1 m × 0.150 m in dimension and was mounted on a shock table. This model, when wrapped with GFRP, was tested under cyclic loading. The lateral cyclic load was transferred from the pendulum impactor. A layer of unidirectional Glass FRP (E-glass - GFRP270 fiber system) was used to wrap the specimen. Thickness of the GFRP sheets was 1.8 mm, and Young's modulus was 70,000 N/mm². The ultimate tensile strength of GFRP was 2,400 N/mm² and the ultimate tensile strain was 0.02.

The stiffness of the SPRING element was equal to that of the GFRP material (70,000 N/mm²). A new concrete model (Concrete Damaged Plasticity Model) in ABAQUS version 6.10 was used to simulate the behaviour of mortar of the walls. This model was a continuum plasticity-based damage model for concrete. It assumed that the two main failure mechanisms are tensile cracking and compressive crushing of the concrete material, and that the uni-axial tensile and compressive response of concrete was characterized by damaged plasticity. Under uni-axial tension, the stress-strain response followed a linear elastic relationship until the failure stress to σ, which represented the onset of micro-cracking in the concrete material. Beyond this failure stress, the formation of micro-cracks was represented macroscopically with a softening stress-strain response.

Under cyclic loading conditions the degradation mechanisms were quite complex, involving the opening and closing of previously formed micro-cracks and their interaction. The stiffness recovery effect, namely some recovery of the elastic stiffness as the load changes sign in cyclic test was considered.
6.3 MICRO-MODELLING APPROACH

A detailed micro-modeling approach was selected for the analysis in which bricks and mortar were modelled as continuums connected to each other by brick-to-mortar interfaces consisting of contact elements as shown in Figure 6.1. This approach was selected to investigate both the global behaviour of the wall and the behaviour of the GFRP wraps. The modelling approach was three-dimensional. Non-linear geometric and non-linear material behaviours were considered in this approach.

![Figure 6.1 Micro-model approach](image)

6.4 FINITE ELEMENT MODEL AND MATERIAL PROPERTIES

A brick panel was modelled using eight-node solid elements. It was assumed that the panel behaves nonlinearly under applied displacement. The mortar between the bricks was modelled using interaction element in ABAQUS which allows for little relative slippage in the contact surface. The properties of this element were defined according to the properties of mortar in the test specimens. Steel frame modelling was carried out using shell elements. To obtain accurate results, the meshing of finite element model was fine enough. As the FRP material has been assumed to behave as a linearly elastic and orthotropic material, a “lamina” option for the elastic behavior of the material was chosen.
The model was imported from the hypermesh to ABAQUS. This import didn’t need any import factor as these two softwares use the same units of measurement. The loading was bit tricky in the way that a fixed end loading has no big effect over the model. Hence the loading frame along with the pendulum set up was modelled along with the model. Later, the velocity of impact was introduced into the analysis as the result of which the loading defined. The impact produced shock waves which travel from the bottom to the top of the model. These shock waves got distributed over the model till all the energy due to impact got reduced to zero. An analysis time of 20 micro second after the impact was carried out over each of the model.

ABAQUS incorporated a special model definition. It was nothing but the plasticity damage model which has the incorporation of micro cracking in bricks and mortar. The opening and closing of these micro cracks was also taken into account according to this damage model. The crack propagation and crack growth was taken care by this model definition. The plasticity damage model incorporated the plastic failure which accounts the presence of steel in the model. As the steel component has not been defined in the modeling, it didn’t consider the presence of the combined action of steel and mortar, but only the brick and mortar was considered. Hence this definition of the model suited the actual experimental model in this case.

There are certain constraints in the experimental part that are unable to be incorporated in the analytical part. One major thing was the movement of the frame that cannot be considered in the analytical part as it was made analysis more complicated. The energy dissipated in the movement of the frame after the impact should be excluded in the experimental part or included in the analytical part. Hence the energy used up to move the frame was included in the analysis part by excluding the impact energy that was used to make this movement there by making a counter balance.
6.5 STRUCTURAL MODEL

A three dimensional FE model was constructed for analysis of the masonry wall using a combination of finite elements. An isometric view of the FE model is shown in Figure 6.2. The FE analysis was conducted through implementation of the ABAQUS finite element computer program.

Figure 6.2 Isometric view of the finite element model of Masonry wall

6.6 ABAQUS MODELLING OF MASONRY WALL SPECIMEN

The meshing of plastered panel and panels with different GFPR wrappings are shown in Figure. 6.3 to 6.6.
Figure 6.3 FEM model of plastered masonry wall specimen

Figure 6.4 FEM model of plastered masonry wall specimen with vertical GFRP wrapping
Figure 6.5  FEM model of plastered masonry wall specimen with diagonal GFRP wrapping with outer edges

Figure 6.6  FEM model of plastered masonry wall specimen with inner diagonal GFRP wrapping
PLASTERED MASONRY WALL PANEL OF PEAK RESPONSE ACCELERATION GRAPH

The acceleration – time history of plastered masonry wall panels is shown in Figure. 6.7

Figure 6.7  Peak base acceleration during the first impact conducted on plastered mason panel when pendulum released at 30 cm height

UNPLASTERED MASONRY WALL PANEL OF PEAK RESPONSE ACCELERATION GRAPH

The acceleration – time history of unplastered masonry wall panels is shown in Figure. 6.8

Figure 6.8  Peak base acceleration during the first impact conducted on unplastered masonry panel when pendulum released at 30 cm height
6.9 PLASTERED MASONRY WALL PANEL STRENGTHENED WITH GFRP VERTICAL WRAPPING OF PEAK RESPONSE ACCELERATION GRAPH

The acceleration – time history of plastered masonry wall panels with GFRP vertical wrapping is shown in Figure. 6.9.

![Graph showing acceleration-time history for plastered masonry panel.]

**Figure 6.9** Peak base acceleration during 51st impact conducted on plastered masonry panel using GFRP vertical strips when pendulum released at 45 cm height

6.10 UNPLASTERED MASONRY WALL PANEL STRENGTHENED WITH GFRP VERTICAL WRAPPING ABAQUS - PEAK RESPONSE ACCELERATION GRAPH

The acceleration – time history of unplastered masonry wall panels with GFRP vertical wrapping is shown in Figure. 6.10

![Graph showing acceleration-time history for unplastered masonry panel.]

**Figure 6.10** Peak base acceleration during 51st impact conducted on unplastered masonry panel using GFRP vertical strips when pendulum released at 45 cm height
6.11 PLASTERED MASONRY WALL PANEL STRENGTHENED WITH GFRP DIAGONAL WRAPPING WITH OUTER EDGES OF PEAK RESPONSE ACCELERATION GRAPH

The acceleration – time history of plastered masonry wall panels with GFRP diagonal wrapping with outer edges is shown in Figure. 6.11.

![Graph showing acceleration-time history for plastered masonry panel](image)

**Figure 6.11** Peak base acceleration during 41st impact conducted on plastered masonry panel using GFRP strips diagonal wrapping with outer edges when pendulum released at 45 cm height

6.12 UNPLASTERED MASONRY WALL PANEL STRENGTHENED WITH GFRP DIAGONAL WRAPPING WITH OUTER EDGES OF PEAK RESPONSE ACCELERATION GRAPH

The acceleration – time history of unplastered masonry wall panels with GFRP diagonal wrapping with outer edges is shown in Figure. 6.12

![Graph showing acceleration-time history for unplastered masonry panel](image)

**Figure 6.12** Peak base acceleration during the first impact conducted on unplastered masonry panel using GFRP strips diagonal wrapping with outer edges when pendulum released at 30 cm height
6.13 PLASTERED MASONRY WALL PANEL STRENGTHENED WITH GFRP INNER DIAGONAL WRAPPING OF PEAK RESPONSE ACCELERATION GRAPH

The acceleration – time history of plastered masonry wall panels with GFRP inner diagonal wrapping is shown in Figure. 6.13.

![Graph of plastered masonry wall panel acceleration](image)

**Figure 6.13** Peak base acceleration during 51st impact conducted on plastered masonry panel using GFRP strips with inner diagonal wrapping when pendulum released at 45 cm height

6.14 UNPLASTERED MASONRY WALL PANEL STRENGTHENED WITH GFRP INNER DIAGONAL WRAPPING OF PEAK RESPONSE ACCELERATION GRAPH

The acceleration – time history of unplastered masonry wall panels with GFRP inner diagonal wrapping is shown in Figure. 6.14.

![Graph of unplastered masonry wall panel acceleration](image)

**Figure 6.14** Peak base acceleration during the first impact conducted on unplastered masonry panel using GFRP strips with inner diagonal wrapping when pendulum released at 30 cm height
6.15 SUMMARY

This chapter presents the numerical investigation of un-plastered and plastered virgin panels as well as those strengthened with GFRP wrapping with different patterns using ABAQUS software. Micro-modelling approach was explained to simulate the interface between brick and mortar. A new concrete model (Concrete Damaged Plasticity Model) in ABAQUS version 6.10 was used to simulate the behaviour of mortar of the walls. The model was imported from the hypermesh to ABAQUS. A three dimensional FE model was constructed for analysis of the masonry wall using a combination of finite elements. The masonry model was 1 m × 1 m × 0.150 m in dimension and was mounted on a shock table. The meshing models of plastered and unplastered panels and panels with different GFPR wrappings are shown in the chapter. The results of analysis in terms of time-history of acceleration of different meshed panels are also presented.