Chapter 5

Finite Element Formulation

5.1 Introduction:

Seismic behavior of infilled RC frames has not been found to be satisfactory during several past earthquakes, and this may be due to modification of the global response of the structure due to the decrement of the natural period, soft-storey sway mechanism etc. Adequate comprehension of the behavior of infilled RC frames is a vital either for designing seismic resistant structures or for repairing and retrofitting existing buildings. Thus, adequate knowledge of the seismic response is very important to reduce the loss of life and property damage, associated with the failure of the infilled RC frames.

Finite element modeling is frequently used to overcome experimental limitations in predicting and analyzing the performance of structures. Different analytical models have been used to describe the behavior of infilled RC frames. The finite element method has been extensively used for modeling infilled RC frame structures, since Mallick and Severn [37] applied this approach in 1967. These models can be divided into two groups; micro-models and macro-models. The finite element formulation and the equivalent truss mechanism are the typical examples of the first and second group, respectively. Macro-models exhibit obvious advantages in terms of computational simplicity and efficiency. Their formulation is based on a physically reasonable representation of the infilled RC frame. The single strut model is a simple representative, but is not able to describe the local effects occurring in the surrounding frame. The use of multi-strut models can overcome this problem without a significant increase in the complexity of the analysis. Palyakov [44], Stafford smith [53, 54, 55], Paulay and Priestly [41] have extensively used equivalent strut model.

FE-models can simulate the structural behavior with great detail, providing that adequate constitutive models are used. Suyamburaja Arulselvan et al [58], Yasar Kaltakei et al [60], Li B et al [33] have studied the behavior of infilled RC frames under seismic load by finite element analysis using Finite Element Software ANSYS. However, they are computationally intensive and difficult to apply in the analysis of large structures. Due to the composite characteristics of infilled RC frames, different elements are required in the model. Finite element models exhibit obvious advantages for describing the behavior of infilled RC frames and the local effects. This implies a greater computational effort and
more time in preparing the input data and in analyzing the results. It is worth noting the importance of defining the constitutive relationships of the elements used in modeling. Even though it is commonly considered that the use of two dimensional elements leads to acceptable results in this study three dimensional elements are used to get a more realistic approach to the complicated behavior of infilled RC frames.

5.2 Scope of the Study:

Dynamic response of Reinforced rat-trap bond masonry infilled RC frames, Plain masonry infilled RC frames and Bare RC frames by considering various parameters such as shape and size of infill panel and size of frame members are to be studied. The behaviour of frames with stilt floors is to be studied with different types of infills. It is proposed to use locally available clay bricks with reinforcement to increase the ductility and seismic resisting capacity of the RC infilled RC frame. FEM software ANSYS is proposed to be used for the finite element modelling and analysis using three dimensional elements.

5.2.1 Number of storeys

Majority of buildings in urban areas of our country are not more than four storey height. In the present investigation the floor to floor height is maintained at 3.0 constant and makes the panel height to vary from 2.5m to 2.6m depending on the size of the beam while the span is varying in the range of 4m to 5m. The beam and column dimensions are varied as per span length.

5.2.2 Shape and size of infill panel:

The infill panels are usually square or rectangular depending on the type of building and spacing of columns. Panel aspect ratio is the ratio of length of the infill panel to its height. In this study, three cases of aspect ratios are chosen. The variation in aspect ratio is obtained by varying the length and height of the infill panel. The thickness of walls may vary from 100 to 230 mm. The thickness of the masonry wall is kept constant as 230mm. Table 5.1 shows the details of the frames considered in this study.

![Fig.5.1: Panel Dimension and Frame Member Cross Section](image-url)
Table 5.1 Details of the frame.

<table>
<thead>
<tr>
<th>Aspect Ratio</th>
<th>Panel size (mm) (l x h)</th>
<th>Frame Member sizes (mm)(b x D)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.23</td>
<td>3200 x 2600</td>
<td>230 x 400</td>
</tr>
<tr>
<td>1.41</td>
<td>3600 x 2550</td>
<td>230 x 450</td>
</tr>
<tr>
<td>1.60</td>
<td>4000 x 2500</td>
<td>230 x 500</td>
</tr>
</tbody>
</table>

5.2.3 Diameter and Spacing of masonry reinforcement:

Mild steel bars of 10 mm diameter are used as masonry reinforcement. The minimum spacing of reinforcement available for Rat-trap bond masonry available is 175 mm. However, a spacing of 400, 450 and 500 mm is adopted for both horizontal and vertical reinforcement in panels of aspect ratio 1.23, 1.41 and 1.60 respectively.

5.2.4 Type of masonry infill:

The following are the different type of infill masonry considered for the study.
1. Plain masonry in English bond (EBI).
2. Rat-trap bond Masonry with grout (GRTI)
3. Reinforced rat-trap bond masonry with horizontal reinforcement (GRTIH)
4. Reinforced rat-trap bond masonry with vertical reinforcement (GRTIV)
5. Reinforced rat-trap bond masonry with both horizontal and vertical reinforcement (GRTIB)

5.2.5 Material properties:

Properties shown in Table 5.2 are considered for different materials in the finite element model.

Table 5.2 Material Properties

<table>
<thead>
<tr>
<th>Material</th>
<th>Young’s Modulus (N/mm²)</th>
<th>Poisson’s Ratio</th>
<th>Density (kN/m³)</th>
</tr>
</thead>
<tbody>
<tr>
<td>M25 concrete</td>
<td>25,000*</td>
<td>0.20</td>
<td>25.00</td>
</tr>
<tr>
<td>Grouted Rat-Trap bond masonry</td>
<td>3,285**</td>
<td>0.15</td>
<td>20.00</td>
</tr>
<tr>
<td>English Bond masonry</td>
<td>1,566**</td>
<td>0.15</td>
<td>20.00</td>
</tr>
<tr>
<td>Reinforcement steel</td>
<td>2 x 10^3***</td>
<td>0.30</td>
<td>78.50</td>
</tr>
</tbody>
</table>

* As per IS: 456:2000 [67], ** From experimental program. *** As per IS: 432:1982 [75].
5.2.6 Earthquake Loads:

The three seismic load cases considered for ground motion,


iii) Load case 3. Kobe Earthquake Data

5.3 The Finite Element Model:

5.3.1 Introduction:

Finite Element Software ANSYS 10 is to be used for the finite element modeling and analysis. Finite element technique is a powerful and versatile tool for the analysis in structural and continuum mechanics. Proenca et al [45] have reported that proper modeling methodology has to be adopted with widely disseminated Finite Element Method programmes with sound engineering judgment.

5.3.2 Modeling:

The element types used in developing the model for infilled RC frames are shown in Table 5.3 and fig.5.1 to 5.3.

<table>
<thead>
<tr>
<th>Material type</th>
<th>ANSYS Element</th>
</tr>
</thead>
<tbody>
<tr>
<td>Concrete</td>
<td>Solid 65</td>
</tr>
<tr>
<td>Masonry</td>
<td>Solid 45</td>
</tr>
<tr>
<td>Steel reinforcement</td>
<td>Link 8</td>
</tr>
</tbody>
</table>

SOLID65 allows the presence of four different materials within each element; one matrix material (e.g. concrete) and a maximum of three independent reinforcing materials. The concrete material is capable of directional integration point cracking and crushing besides incorporating plastic and creep behavior. The reinforcement (which also incorporates creep and plasticity) has uni-axial stiffness only and is assumed to be smeared throughout the element. Directional orientation is accomplished through user specified angles. Solid65 is used for the 3-D modeling of solids with or without
reinforcing bars (rebar). The solid is capable of cracking in tension and crushing in compression. In concrete applications, for example, the solid capability of the element may be used to model the concrete while the rebar capability is available for modeling reinforcement behavior. The element is defined by eight nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions. A schematic diagram of the element is shown in Figure 5.2.

![Figure 5.2 – Solid 65 Element](image)

SOLID45 is used for the 3-D modeling of solid structures. The element is defined by eight nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions. The element has plasticity, creep, swelling, stress stiffening, large deflection, and large strain capabilities. The geometry and node locations for this element are shown in Figure 5.3.

![Figure 5.3 – Solid 45 Elements](image)

LINK8 is a spar which may be used in a variety of engineering applications. This element can be used to model trusses, sagging cables, links, springs, etc. A schematic diagram of the element is shown in figure 5.4. The 3-D spar element is a uni-axial
tension-compression element with three degrees of freedom at each node: translations in the nodal x, y, and z directions. As in a pin-jointed structure, no bending of the element is considered. Plasticity, creep, swelling, stress stiffening, and large deflection capabilities are included. The element is defined by two nodes, the cross-sectional area, an initial strain, and the material properties. The element x-axis is oriented along the length of the element from node i towards node j. The initial strain in the element is given by $\Delta/L$, where $\Delta$ is the difference between the element length, L, (as defined by the i and j node locations) and the zero strain length. Fig 5.5 shows the finite element model of a typical storey.

In the present investigation, RC frame models considered have spans of 4.0m, 4.50m & 5.00m and floor to floor height of 3.0m. The model in each case is a 4 storey RC frames with various types of infill. FE models are created using solid 65 concrete
elements for frame sections, solid 45 brick element for infill masonry and the reinforcement is modeled using Link 8 element.

5.3.3 Meshing:

For better convergence of results from the Solid 65 and Solid 45 elements the element aspect ratio has been maintained at 1.0 by varying the mesh size for the aspect ratios of 1.23, 1.41 and 1.6 as per the following

- 16x13 elements for aspect ratio 1.23
- 16x12 elements for aspect ratio 1.41
- 16x10 elements for aspect ratio 1.6

The bounding frame is represented by Solid 65 reinforced concrete element which in which reinforcement is represented by smeared layer. Solid 45 element representing masonry has 8 nodes. Additional nodes are created while meshing in the middle of thickness, which are connected by Link 8 elements representing steel reinforcement. The reinforcement is taken through the frame members. Strain ratio between the concrete and steel is supposed to be equal, therefore it is accepted that there is a unique adherence between the concrete and steel. However, the necessary mesh attributes of the reinforcement are created. The nodes of reinforced concrete element, the corresponding nodes of masonry and steel are merged. Caution is taken while merging entities in a model. Fig 5.6 shows the finite element discretization.

![Finite Element Discretization](image)

Fig.5.6: Finite Element Discretization
5.4 Analysis:

5.4.1 Response Spectrum Method:

Response spectrum analysis is performed on the models by captivating the Response spectrum data from IS. 1983-2002 [65] for zone 4 and zone 5, having the soil type III and by taking the Kobe Earthquake data. The graphical representation of the earthquake analog input data of Zone IV, Zone V and Kobe earthquake have been presented in figures 5.7.

![Fig. 5.7 Typical Graph of Period vs Acceleration](image)

The response spectrum analysis is carried out in two steps

- Modal Analysis
- Spectral Analysis

5.4.1.1 Modal Analysis:

Modal Analysis is the study of the natural characteristics of structures. This analysis characterizes the dynamic properties of an elastic structure by identifying its mode of vibration. Modal analysis in the ANSYS family of products is a linear analysis. The response of the structure is different at each of the different natural frequencies. These deformation patterns are called mode shapes. Both the natural frequency (which depends on the mass and stiffness distribution in structure) and mode shape are used to help the design of structural system mainly for dynamic applications. Enough modes are to be selected to cover the frequency range spanned by the spectrum and to characterize the structure's response. The accuracy of the solution depends on the number of modes used: the larger the number, the higher the accuracy. In the present work 10 mode shapes have been evaluated. The modal solution is required because the structure's mode shapes and frequencies must be available to calculate the spectrum solution. Also, by performing the spectrum solution ahead of mode expansion, one can expand only the significant modes that contribute to the final solution.
5.4.1.2 Spectral Analysis:

A response spectrum represents the response of single-DOF systems to a time-history loading function. It is a graph of response versus frequency, where the response is acceleration. Single-point response spectrum method is used for the present work. Only linear behaviour is valid in a spectrum analysis.

In a single-point response spectrum (SPRS) analysis, specification of one response spectrum curve is given at a set of points in the model, such as at support.

Excitation at the support (base excitation) and excitation away from the support (force excitation) both are allowed for the single-point response spectrum analysis. In the present investigation base excitation at the support has been employed.

5.4.1.3 Modal combination:

Structural design is usually based on the peak values of forces and deformations over the duration of the earthquake induced response. The peak response quantities can be combined as per square-of-sum-of-squares (SRSS) rule. Only those modes having significant amplitude (mode coefficient) are chosen for mode combination. The peak response in each mode is squared, the squared modal peaks are summed, and the square root of the sum provides an estimate of the peak total response. This modal combination rule provides excellent response estimates for structures. In this study SRSS rule is used to evaluate the peak total response.

5.5 Conclusion:

Finite element formulation is made using appropriate elements and it is expected that these models will simulate the infilled RC frames efficiently and be used to study the stress and deformation responses of infilled RC frames for earthquake induced ground motions, which is discussed in the next chapter.