IF : 3.62 | IC Value 70.36

Volume-5, Issue-11, November - 2016 • ISSN No 2277 - 8160

Original Research Paper

Engineering

# Experimental and Analytical Study on 2D Rc Frame with Ferrocement Infill

N.Jayaramappa

Associate Professor, Department of Civil Engineering UVCE, Bangalore University, Gnanabharathi Campus, Bangalore 560056

**ABSTRACT**

*This paper presents an experimental investigation on the behaviour of a 2D single bay two storey reinforced concrete (RC) frame with ferrocement panel as infill. RC bare frame is cast in the laboratory to a scale down of 1:3.25; the dimension of frame is 2300 mm height and 1000 mm width. The cross section of beam and column are 100mm x 70 mm. Ferrocement panel is prepared for 1000 mm x 1000mm x 25mm, the panel was placed in to the bounding frame with cement mortar . The proposed model is subjected to lateral load at each storey level and their performance was assessed based on load carrying capacity and deflection. The present study includes the entire range of loading from the initial elastic stage until the ultimate failure stage. Analytical study was also conducted for the similar RC frame with ferrocement panel infill analytical results were obtained using finite element analysis software ETABS. Analytical results were compared with experimental results and it was found to compare well.*

**KEYWORDS : Ferrocement, ETABS, Pushover Analysis and lateral load**

## 1. INTRODUCTION

Ferrocement has been developed during the past fifty years and yet has reached a very advanced stage in technique and design. A considerable amount of laboratory testing research and prototype constructions have been completed at research centers, the production of ferrocement members that would be used in the roof ,floor, walls of building, domes, sculptures, chajjas etc .

It consists basically a matrix of mortar with high proportion of steel reinforcement in the form of wire mesh. It is relatively cheap, strong and durable, and the basic technique is easily acquired. Ferrocement has a great strength and economy. It is fireproof, earthquake safe and does not rust. A working definition of Ferrocement is " a thin shell of highly reinforced Portland cement mortar" Generally, Ferrocement shells range from 10 to 50 mm in thickness and the reinforcement consists of layers of steel mesh; usually with steel reinforcing bars sandwiched midway between other materials such as selected organic, natural or synthetic fibers may be combined with metallic mesh, admixtures super plasticizers, fibers etc. The high quality cement matrix has a ratio of 1:2 to 1:2.5 or so. Over this finished framework, an appropriate mixture of cement, sand and water is spread out. During hardening, the Ferrocement is kept moist, to ensure the cement is able to set and harden.

## 2. EXPERIMENTAL WORK

The experimental work consists of the following steps.

- Casting of 2D RC bounding frame.
- Casting of Ferrocement infill wall within the RC frame.
- Testing arrangement.
- Testing of Infilled frame under progressive lateral load and measure of deformations.

### 2.1 Casting of 2D RC bounding frame.

RC bare frame is cast in the laboratory to a scale down model of 1:3.25; the dimension of frame is 2300 mm height and 1000 mm width, the cross section of beam and column are 100mm x 70 mm, figure (1) shows the details of 2D RC frame model. The concrete mix is designed as per IS: 10262-1982 for a characteristic strength of 20N/mm$^2$. After 28 days of curing period was over ,the frame is lifted and transported in to the loading frame with the help of the overhead crane.

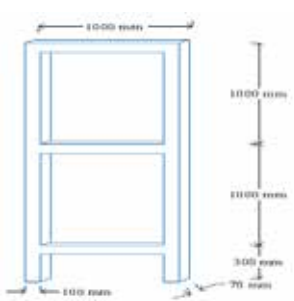

**Fig.1. Experimental model**

### 2.2 Casting of Ferrocement panel within the RC Frame.

The following steps are followed to cast Ferrocement panel infill.

Wooden formwork is prepared using 12 mm thick ply wood sheet of 1000 mm length, 1000 mm width and 70 mm thick with suitable stiffeners . the details are shown in figure (2). Prepare reinforcement cage for panel with 6 mm dia steel rods are at 250mm x250mm for the size 800 mm x800 mm in both the direction. Cut the chicken mesh for the size 960 mm X 960 mm to fit into the mould with clear cover of 20 mm. Weigh the required amount of the cement, fine aggregate, water and super plasticizer according to the mix design. Place the weighed cement and sand in the dry clean pan mix it in dry condition and mix them thoroughly. Once the dry mix is ready add water and SP, mix the mortar thoroughly. Place the mixed mortar in the panel about a height of 10 mm, tamp it with the tamping rod (25 times) and level it. Place the chicken mesh on the layer of the mortar as shown in figure (2), pour the mortar for the remaining height of 10 mm and do the leveling and compaction. Allow the panels so prepared to set for 24 hours and de-mould the panels. Figure (3)shows the completed ferrocement panel.

**Table (1). Details of RC frame with ferrocement panel infill**

| Parameter | | Frame |
|---|---|---|
| Type of frame | | 2D |
| Number of bays | | 1 |
| Number of story | | 2 |
| Bay length | | 1000 mm |
| Storey height | | 1000 mm |
| Structural material | | Reinforced concrete |
| Concrete | Mass | 24.12kN/m$^3$ |
| | Compressive strength [MPa] | 26.60N/mm$^2$ |
| | Modulus of elasticity supposed [MPa] | 24556.05 N/mm$^2$ |
| | Poisson Ratio | 0.2 |
| Rein-force-ment. | Modulus of elasticity supposed [MPa] | 210000 |
| | Yield strength $f_y$ [MPa] | 415 |
| Column. | Section length [mm] | 100 |
| | Section width [mm] | 70 |
| | Reinforcement | 4 Ø 8 |
| | Percentage of steel | 2.87 |
| Beam. | Section height [mm] | 100 |
| | Section width [mm] | 70 |
| | Reinforcement<br>Percentage of steel | 4 Ø 8<br>2.87 |

| | | |
|---|---|---|
| Ferrocement | Compressive strength [MPa] | 49.04 (% reinforcement) |
| | Mass | 22.13 kN/m$^3$ |
| | Modulus of elasticity supposed [MPa] | 26700N/mm$^2$ 26.60N/mm$^2$ |
| | Poisson ratio | 0.22 |
| | Ductility ratio | 3.37 |

**Fig.2 Placing the reinforcement inside the formwork for Ferrocement panel**

**Fig.3 Ferrocement panel**

**Fig.4. RC frame with ferrocement panel infill**

### 2.3. Testing Arrangement

The RC frame was cast in the laboratory and sufficient precautions are taken so that the specimen could be easily removed from the casting place and erected. Then the frame was lifted and transported to the testing block with the help of the overhead crane . The testing frame is placed in to the lateral loading frame. Fixity is achieved by applying Nitobond and clamping using MS clamp plate as shown in figure (4). Two loading points were selected at first and second storey levels. The load points roughly stimulate the equivalent static seismic load to the frame. The static lateral incremental loads were applied at the jack locations to the frame by hydraulic jacks of 500kN capacity with least measurable value of 2.5kN. The jacks are placed horizontally in line with centre of beams; horizontality of jacks is confirmed using spirit level. The loading frame, which is used for loading arrangements, is rigidly fixed to the floor. The jacks are fixed to the loading frame. Load is transferred to the specimens in the form of uniformly distributed load pattern; the jacks were controlled by an individual console. For the application of load, hand operated oil pumps were used. Crack pattern and displacements are observed at all increment of loading. The apparatus used are as follows.

1. 50kN capacity Lateral loading frame.
2. 2D RC Frame with brick infill (Testing frame).
3. LVDT (0.01mm three dial gauges) instrument.
4. 2 Hydraulic jacks of 500kN capacity of 2.5 kN least measurable value.

### 2.4. Testing of Infilled frame under progressive Lateral Load and Measure of Deformations

The lateral loads are applied at the 1st and 2nd storey level using hydraulic jacks. The load increment for each interval is 2.5kN. The first crack in mortar joint was observed at the total load ($P_1$+$P_2$) of 25.75 kN for a deflection of 23.38mm, details are shown in table (2). Gradually load is increased for same intervals. The application of load is stopped at 37.5kN for deflection of 47.31mm, as the cracks widened and no further new cracks have developed.

LVDTs (Linear Variable Differential Transformers) of least count 0.01mm are used for measuring displacement at each storey level. LVDTs were connected to slotted angles that are in turn connected to the fixed type of steel reaction frame available in the laboratory. The load increment for each cycle is 2.5kN. The deflections at all storey levels were measured using LVDT at each increment of load. The load increments are continued till the final cracks occurred in all joints.

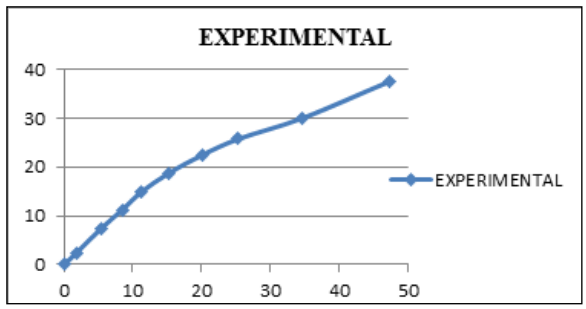


**Fig. 5 Base Shear V/S Displacement Curve for Experimental Results.**

**Fig.6. Crack appear in column compression Side.**

### 3. ANALYTICAL STUDY

Analytical study is also conducted for the similar RC frame with Ferrocement panel infill ; analytical results were obtained using finite element analysis software ETABS -9.7.1. Analytical results were compared with experimental results.

To compare experimental results with analytical results, frame with Ferrocement panel infill models are prepared and analyzed using finite element package ETABS (version-9.7.1). It is versatile and user-friendly software that offers a wide scope of features like static, dynamic, linear and nonlinear analysis etc. Analytical results are compared with experimental results.

**Modeling of Ferrocement panel Infilled Frame**

Ferrocement panel have significant effect in stiffness, strength and seismic performance of buildings. In case of uniformly Infilled frame buildings, strength capacity increases than that of bare frame buildings with the reduction in displacement.

The total storey shear force increases considerably as the stiffness of the building increases in the presence of Ferrocement panel (figure 6a &6b). The mode of failure in soft storey mechanism (formation of hinges in ground floor columns), the lateral load resisting mechanism of the infilled frame are essentially different from the bare frame. The bare frame acts primarily as a moment resisting frame with the formation of plastic hinges at the joints under lateral loads. In contrast, the infill frame behaves like a braced frame resisted by a truss mechanism formed by the compression in the Ferrocement panel infill and tension in the column.

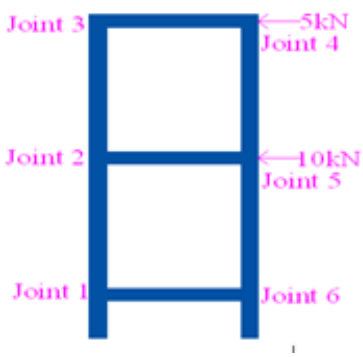


**Fig .6a Details of basic Analytical model**

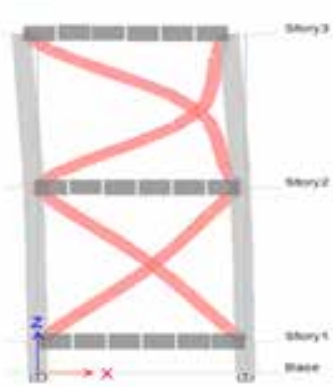

**Fig.6b Details of analytical model with diagonal struts**

**3.2 Geometric Modeling (modeling of frames)**

ETABS 9.7.1 offers an option to choose 2D and 3D geometric models, depending upon the user's convenience and problem definition, A 2D model of the RC frame is developed . Beams and columns are modeled by frame element formulation. Member stiffness is defined by the dimensions of the section, assigned through section properties and modulus of elasticity of the concrete. The details of the structure are shown in table (3).

**Material Properties**

The material properties considered for the analysis are given below. Material Characteristic strength (MPa) of Concrete (M20) $fck = 26.44$ N/mm$^2$ $Ec = 25709.920$ N/mm$^2$Modulus of Elasticity (MPa) of Reinforcing steel $fy = 415$N/mm$^2$ $Es = 2$ E+5 MPa, Characteristic strength (MPa) of Ferrocement panel $ffc = 49.04$ N/mm$^2$ $Efc = 26700.00$ N/mm$^2$, Poisons ratio=0.22.

**3.4 Structural Modeling**

The analytical model is created in such a way that the different structural components represent as accurately as possible the characteristics like mass, strength, stiffness and deformability of the structure.

**(a) Beams and columns:** Beams and columns are modeled as 3D frame elements. The members were represented through the assigned properties like cross sectional area, reinforcement details and the type of material used.

**(b) Beam-column joints:** The beam-column joints were assumed to be rigid and modeled by giving end-offsets to the frame elements. The intension is to get the bending moments at the face of the beams and columns. A rigid zone factor of -1 is considered to ensure rigid connections of the beams and columns.

**(c) Plastic Hinge:** When a concrete element undergoes large deformations in the post-yield stage, it is assumed that the entire deformation takes place at a point called "plastic hinge". The hinges represent concentrated post yield behaviour in one or more degree of freedoms. Each plastic hinge is modeled as a discrete point hinge. Hinges can only be introduced in frame elements at any location. ETABS implements the plastic hinge properties described in FEMA-356 (or ATC-40).

ETABS also gives the choice for uncoupled moment (M), torsion (T), axial force (P) and shear (V) hinges and coupled P-M3, P-M2 and P-M2-M3 hinges(CSI Analysis reference manual), which yields based on the interaction of axial force and bending moments at the hinge location. More than one type of hinge can exist at the same location, for example, both M3 (moment) and V2 (shear) hinge can be assigned to the same end of a frame element.

Default and user-defined plastic hinge options are available in ETABS. User-defined hinges are better than the default-hinges in reflecting nonlinear behaviour compatible with the element properties. However, if the default-hinge is preferred due to simplicity, the user should be aware of what is provided in the program. The definition of user-defined hinge properties requires moment–curvature analysis of each element. For the problem defined, building deformation is assumed to take place only due to moment under the action of laterally applied earthquake loads. Thus user-defined M3 hinge is assigned at member ends where flexural yielding is assumed to occur.

**3.5 Pushover Analysis**

The following steps are followed to static non-linear Pushover analysis of Brick Infill frame.

1. ETABS provides a multi unit options, such as lb-ft, Kip-in, KN-cm, Kgf-mm, T-m, KN-m out of which KN-m unit is selected to create the basic model.
2. Go to file menu click on the new dialog box to create a new model, and select the default edb to select a model type.
3. Select uniform grid spacing option, the uniform grid spacing option requires data in the following format. Enter the data as far with experimental model as shown in figure (4)
4. Select grids only menu, press 'OK' button.
5. Grid of the model is created as shown in figure (6a & 6b).
6. Reinforcement details of column and beam are also included.
7. Go to Define menu and define the properties of frame section like beams and column with their respective cross section and reinforcement details.
8. Keep LHS window in Plan section, go to storey 2, go to draw tool bar and select Draw Lines option, dialogue box will appear select the given beam section and draw the line on the selected storey 2, the beam will be assigned and repeat for all storey .
9. Keeping LHS window in Plan section again, go to the storey 2, go to draw tool bar and select Create Columns in Region option, dialog box will appear select the required cross section and click on the selected storey 2 at each corners with respective cross section and columns will be displayed, similarly repeat the procedure for the remaining levels.
10. Keep LHS window in Plan section, go to the base and select all the corners and go to Assign option select joint/point, restraints

and assign the fixity as per the requirement the base will be fixed.

11. Go to Select option, Select By Frame Section Select all Column, then go to Assign option Frame Non Linear Hinges Dialog box will appear drag down Select P- MM 0 and once again P-MM 1. Click on add.
12. Go to Select menu select By Frame Section Select all Beam, then go to Assign option Frame Non Linear Hinges Dialog box will appear drag down Select M3 0 and once again M3 1.Click on add.
13. Now Select RHS window go to the top and click on Set Building View Option, then click on Fill Objects and Extrusion option you will get the model as below
14. Now go to Define menu, select Static Load Cases, here provide 1st load case as Dead Dead Self Multiplier as 1 and Auto Lateral Load is Blank, similarly give 2nd load case as EQ Quake Self Multiplier as 0 and Auto Lateral Load for this we have to mention the code as IS 1893 2000 since the model is analyzed by this code provision of Earthquake, then go to modify and input the data like Zone factor coefficient, Soil type, Natural time Period T, Importance Factor I, Reduction Factor R, and then click on OK (figure 7&8).
15. 16: Now go to Analyze menu, select Check Model, a dialogue box will appear then select all the options and click on OK, model will be checked and warnings will be viewed if any errors are detected.
16. Now again go to Analyze menu, select Run Analysis, and analy-


**Fig.7. Static load Case**

17. After analysis is complete, go to Options menu Preferences Concrete Frame Design, a dialog box will appear and in Design code drag down select Indian IS 456-2000, this is for the design of the RC frame.
18. Now go to Design menu, select Concrete Frame Design select Start Design/Check Structure, the model will be checked for the adequacy of the members according to the IS 456-2000 code and results can be viewed in Display Design Info..
19. Once the design part is over Non linear analysis begins. Go to Define menu, select Non linear/pushover cases Add New Case Case Name as Push 1, set the Push to Displacement Magnitude to the Target displacement and use this target displacement for other models to analyze, select the Number of Total Steps, Maximum Null Steps, then select P-delta in nonlinearity effects, in Load Patterns select Acc in x Direction Scale Factor is -1 then OK.
20. After defining the Non linear cases go to the Run Analysis option select Run Analysis, once this is complete again go to run analysis menu select run Static Non Linear Analysis, analysis will be carried out and after the completion, press OK.
21. Now go to the display menu and select show Static Pushover curve, where we will get the Push over curve and Capacity-Demand Curve as Shown below, again go back to Display and Show Deformed Shape for Push 1 at diff mode shapes we will get the hinge levels as shown below (Figure 8 &9).

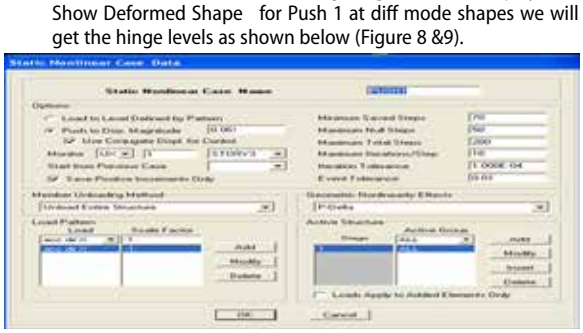

**Fig. 8. Pushover Load Case details**

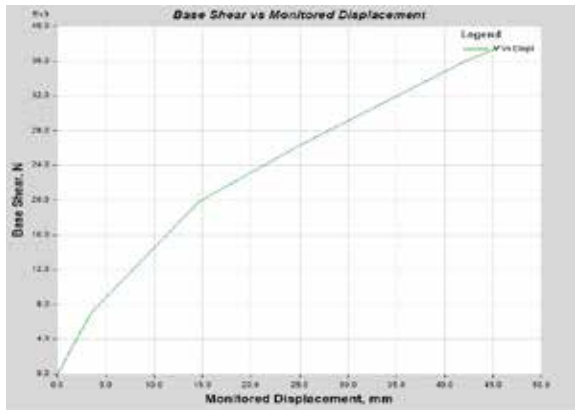


**Fig.9. pushover curve obtained for RC frame with Ferrocement panel infill**

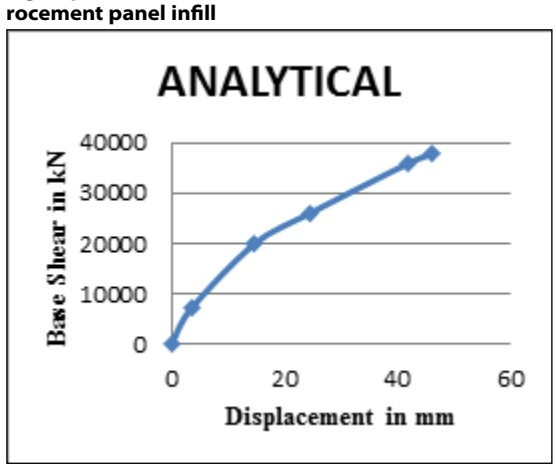


**Fig.10 Base Shear V/S Displacement Curve for Analytical Results**

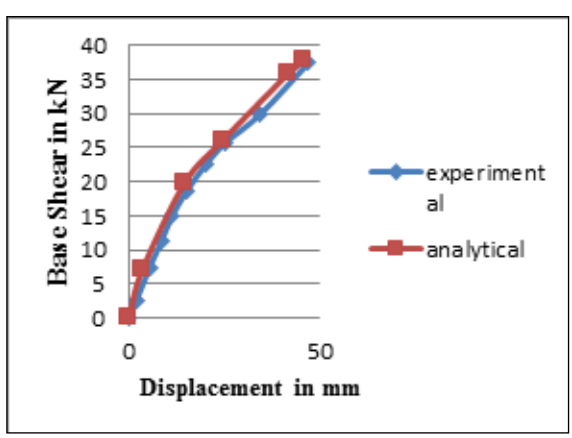


**Fig.11. Comparison analytical and experimental**

**4. RESULTS:**

Two dimensional RC frame is cast and filled with Ferrocement panel infill, the model is tested for lateral load and Analytical study is also conducted for the similar RC frame with Ferrocement panel infill, analytical results were obtained using finite element analysis software Etabs -13. Analytical results were compared with experimental results, table (2) shows the experimental and analytical results, the details are as follows.

**Experimental results.**

Horizontal loads are applied at all storey levels, displacements are measured using LVDT fixed at each storey level. The results are tabulated in table (2). Pushover curve is plot to the above results, details are shown in fig.5. Fig 6 show the cracks at joints 1.

**Analytical results .** Similar models are created using ETABS, deflections obtained in experimental results are put as displacement in pushover analysis, details are shows in fig. (7&8) base shear against displacements are recorded, the details of results are shown in table (3). Fig. (9 &10) shows the pushover curve for analytical results and fig. 11 shows comparison of experimental and analytical pushover curves.

**Table.2 Experimental and Analytical Results of Frame with Ferrocement panel infill**

| Sl. No. | Experimental | | | | | Remarks |
|---|---|---|---|---|---|---|
| | Load | | | Displacement | | |
| | P1 ( kN) Top | P2 (kN) Bottom | Total (kN) | D1 (mm) Bottom | D2 (mm) Top | |
| 1 | 2.5 | 0 | 2.50 | 01.54 | 1.860 | |
| | 5.0 | 2.5 | 7.5 | 2.134 | 5.48 | |
| 2 | 7.5 | 3.75 | 11.25 | 4.45 | 8.490 | |
| 3 | 10 | 5.0 | 15.00 | 6.85 | 11.350 | |
| 4 | 12.5 | 6.25 | 18.75 | 8.13 | 15.270 | |
| 5 | 15 | 7.5 | 22.50 | 13.00 | 20.160 | |
| 6 | 17.5 | 8.25 | 25.75 | 16.010 | 25.380 | Minute First Crack at contact |
| 7 | 20 | 10.0 | 30.00 | 17.02 | 34.680 | Final Column Crack @ Compression |
| 8 | 25 | 12.5 | 37.50 | 26.05 | 47.31 | Final column Crushing |

## 5. DISCUSSION AND CONCLUSIONS.

In the present study 2D RC bare frame with Ferrocement panel infill was subjected to lateral loads and the obtained results were compared with analytical results obtained by ETABS software on similar model. The obtained values of base shear and deflections by experimental and analytical results are compared, the experimentally obtained values are found to be within permissible limit.

## References:


1. L. Landi, P.P. Diotallevi & A. Tardini (2012),"Calibration of an Equivalent Strut Model for the Nonlinear Seismic Analysis of Infilled RC Frames", Department of Civil, Environmental and Materials Engineering - DICAM, University of Bologna, Bologna, Italy .
2. Sachin R Patel , Sumant B patel (May 2011) "Effect of Brick Infill Panel in Design of High Rise Building ",National Conference on Recent Trends in Engineering & Technology, B.V.M. Engineering College, V.V.Nagar,Gujarat,India.
3. K. Chethan, R. Ramesh Babu, Katta Venkataramana and Akanshu Sharma, ( Jun 2010), " Influence of masonry infill on fundamental frequency of 2D RC frames", Journal of Structural Engineering, vol 37,no2 June-July 1010.
4. Ebrahim Khalizadh Vahidi, Maryam Mokhtari Malekabadi, (2009), " Conceptual Investigation of Short- Columns and Masonry Infill Frames Effect in the Earthquake" , World Academy of Science, Engineering and Technology 59.
5. Robin Davis, Devdas Menon and Meher Prsad, (2009), " Earthquake-resistant
6. design of open ground story RC framed buildings", Journal of Structural Engineering, vol 37,no2.