J. Inst. Eng. India Ser. A (February–April 2013) 94(1):23–33
DOI 10.1007/s40030-013-0039-6

ORIGINAL CONTRIBUTION

Finite Element Modeling of Reinforced Concrete Corners Under Opening Bending Moment

N. Bansal · N. Kwatra · V. S. Kanwar

Received: 21 September 2012 / Accepted: 7 October 2013 / Published online: 25 October 2013
© The Institution of Engineers (India) 2013

Abstract In the past few years, there has been growing use of finite element method for analysis of RCC structures to eliminate the expensive testing. However, validations of numerical models are instrumental towards making models reliable, which is, fundamental towards truly predictive prototyping. Keeping this in view, paper presents the results of numerical study of Reinforced Concrete Corners under opening moments using general-purpose finite element analysis software (ANSYS) and the comparison of results to the experimental results available in the literature [Singh and Kaushik, J Inst Eng (India), 84:201–209, 2003]. The Load deflection behaviour obtained is compared with the results corresponding to four different reinforcement detailing investigated experimentally [Singh and Kaushik, J Inst Eng (India), 84:201–209, 2003]. Details of the FE modeling of Reinforced Concrete Corners with four different reinforcement detailing have been presented. The best reinforcement detailing of Reinforced Concrete Corners on the basis of load deflection behaviour has been judged and advocated further on the basis of finite element analysis. The second phase of the study, investigates the effect of diameter of main steel, grade of concrete and spacing of shear reinforcement on load deformation behavior of the corners under opening bending moments.

Keywords Reinforced concrete corners · Finite element analysis · Shear reinforcement · Opening bending moment

N. Bansal
Civil Engineering Department, RIMT-IET,
Mandi Gobindgarh 147301, India

N. Kwatra
Civil Engineering Department, Thapar University,
Patiala 147001, India

V. S. Kanwar (✉)
Civil Engineering Department, Chitkara University,
Solan 174103, India
e-mail: varinderkanwar@gmail.com;
registrar@chitkarauniversity.edu.in

Introduction

In the design of reinforced concrete structures, much of the attention is embarked towards calculation of the strength of basic structural elements like beams, columns and slabs. Comparatively lesser emphasis has been laid on the detailing, corresponding strength and behavior of corner joints, especially those subjected to opening moments as in the case of cantilever retaining walls, bridge abutments, channels, rectangular liquid retaining structures, beam column joints under earthquake loads. The detailing of reinforcement should be easier and simpler in order to expedite the construction process. At the same time structural member should satisfy the fundamental requirements of strength expressed in terms of controlled cracking and ductility.

Investigators have experimentally established the load deflection behaviour of Reinforced Concrete Corners subjected to opening joints. The effect of different detailing systems has been investigated in laboratory but surprisingly, numerically modeling has received lesser attention. For completeness, a brief review of the various studies related to behaviour of opening of Reinforced Concrete Corners joints is included herein.

It has been investigated earlier [1] on the structural behaviour of opening corners reinforced with some of the prevalent and widely used detailing systems. Under the effect of the opening moment four different detailing systems were used. The details of these systems are shown in Fig. 1. The shape and size of the specimens, loading arrangement and instrumentation of the test specimens are illustrated in Fig. 2. All specimens were tested under pure positive (opening) moment using the shown loading arrangement. The specimens were tested in horizontal position (lying on frictionless supports on the ground).

The researchers [2] have carried out finite element analysis of tubular joints to study the fracture mechanics of the joints. A combination of 3-D brick elements near the welded joint and 2-D shell elements in the rest of the regions is suggested computationally efficient. This has been made possible by enforcing appropriate compatibility conditions at the interface between brick and shell elements. It has been concluded that the methodologies for fracture analysis will be useful for the prediction of remaining life of tubular joints, as the prediction of the remaining life essentially depends on the stress intensity factor (SIF) values.

The researchers [3] have theoretically investigated the chamfering in reinforced cement concrete corner subjected to opening moment. They concluded that overall stress level decreases with the increase in chamfer size. They could simulate, using finite element analysis, the increase in the ultimate load carrying capacity is observed in with the increase in chamfer size as observed in the tests.

The literature review indicates the feasibility of finite element modeling of reinforced joint under opening

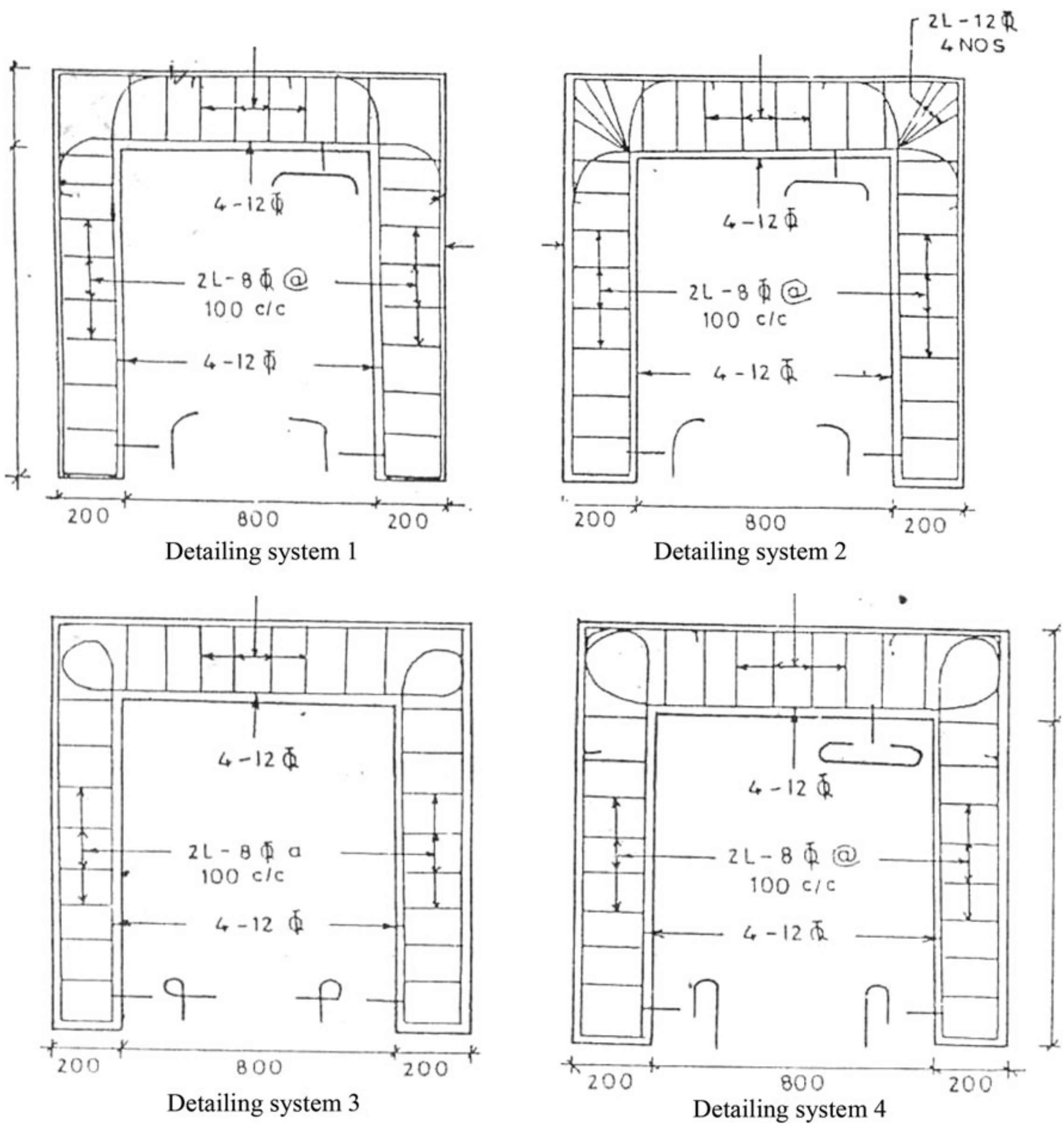
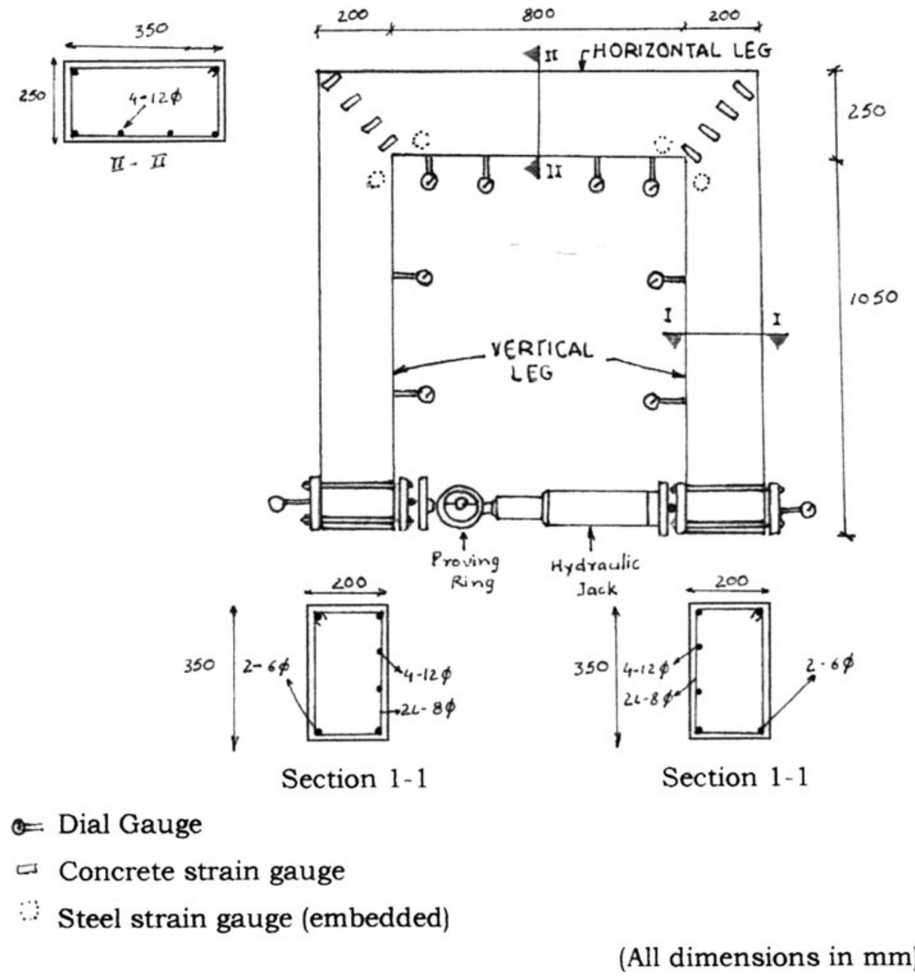


Fig. 1 Detailing of test specimen [1]

Fig. 2 Test set up (plan) [1]



moments. In the present study finite element software (ANSYS) has been used to analyze the Reinforced Concrete Corners. In this study the analysis of the specimens tested in laboratory [1] has been carried out and comparison of the results is presented. The analytical results show good agreement with the experimental results. It is concluded that out of four model **s**pecimens taken for study the specimen having shear reinforcement of spacing 75 mm c/c gives the best results because it takes the maximum load with maximum deflection or it has highest ductility among the three. Gradually on increasing the spacing the ductility goes on decreasing.

Research Significance

Experiments have always played a significant role in the research related to concrete structures. But numerical studies have its own importance, as it can give a much better insight to the behavior of the concrete structures. The complex pattern of stress distributions, which experimental studies can not describe properly, can be captured in a much better way with the help of a successful numerical simulation. One can have a better understanding of the behavior and failure mechanism of the tied columns from the successful numerical investigation and the subsequent parametric study carried out reveals the effects of varying the different parameters. Also, there remains the scope to modify the model so that the parameters remaining uncovered can be included in the simulation, although that is the subject of another work.

Finite Element Modeling

For concrete applications in general, hexahedral elements are found to be more stable and efficient in convergence than the tetrahedral elements. Therefore eight-noded isoparametric element, having translations in x, y and z directions, have been used for the modeling of concrete. A discrete approach, using two-noded three-dimensional isoparametric bar elements, has been adopted for the modeling of reinforcements. The beam column joint has been modeled in finite element analysis software preprocessor using 3D elements (Solid65). Reinforced concrete exhibits inelastic behavior even at early stages of loading and this effect is more pronounced near ultimate load stages. Therefore it becomes essential to carry out a nonlinear analysis, use of proper geometric and material models for both concrete and steel is important.

Concrete exhibits ductility under small hydrostatic pressures and undergo brittle failure in tension. Therefore, a material model able to simulate the behavior of concrete in both tension and compression has been chosen among those available. The actual behavior of concrete in three-dimensional state of stress is extremely complicated. It is well established that under tensile and low compressive stresses, concrete fails by brittle fracture. On the other hand, it can yield and flow likes a ductile material under high hydrostatic pressure.

Several failure models have been developed for brittle and ductile behavior of concrete [4–7]. The most commonly used failure criteria are defined in stress spaces by a number of material constants varying from one to five independent control parameters. Though all these models have certain inherent advantages and disadvantages, yet the William-Warnke five-parameter model, to date, is most versatile and sophisticated criterion for the elasto-plastic modeling of concrete. Another option could have been the Drucker–Prager Model. But, the perfectly plastic assumption of the Drucker–Prager Model fails to capture the dilation characteristics and over-predicts the strength [7].

For modeling concrete in compression, inelastic constitutive relations have been specified by defining yield criterion with the hardening rule and the uniaxial strength. The William-Warnke five-parameter model has been combined with isotropic hardening rule (to describe the changing of the yield surface with progressive yielding) and an associated flow rule (to indicate that the plastic strain would occur in a direction normal to the yield surface) to formulate the inelastic constitutive relations. Thus, the model assumes that, concrete behaves elastically as long as the stress state lies within an initial yield surface. When loading progresses beyond the initial yield surface, plastic flow occurs and the yield surface hardens isotropically up to a failure surface. In this range, the plastic strain rate is governed by the yield function.

Cracking in Concrete

The tension failure of concrete is characterized by a gradual growth of cracks, which join together and eventually disconnect larger parts of the structure. Cracking in matrix aggregate composite like concrete involves micro-cracking, tortuous debonding and the other process of material damage. It is usually assumed that cracking formation is a brittle process and that the strength in tension loading direction abruptly goes to zero after such cracks have formed. Moreover, material like concrete is capable of transmitting stresses due to tortuous debonding and aggregate interlock.

Therefore, the formation of cracks is undoubtedly one of the most important non-linear phenomenon, which governs the behaviour of concrete structures. With the result any computational model, which is to be employed for the analysis of concrete structures must embody a sound numerical procedure that handles the formation of cracks. In the finite element analysis of concrete structures two principally different approaches have been employed for crack modelling. These are (A) discrete crack modelling (B) smeared crack modeling.

The discrete approach is physically attractive (Fig. 3), as it reflects the highly localized nature of cracking. But, this approach suffers from few drawbacks, such as, it employs a continuous change in nodal connectivity, which does not fit in the nature of finite element displacement method; the crack is considered to follow a pre-defined path along the element edges and excessive computational efforts are required. The second approach is smeared crack approach. In this approach the cracks are assumed to be smeared out (Fig. 4) in a continuous fashion. Computationally this approach is much simpler, as only the constitutive relation expressed in terms of stresses and strains needs to be modified in the region of interest. In finite element analysis of reinforced concrete, the development of non-linear procedure has generally progressed along two lines viz., fixed crack modeling and rotting modeling. In the fixed

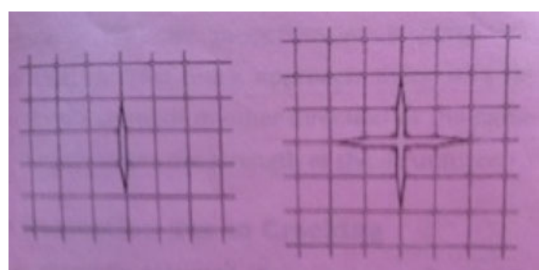


Fig. 3 Discrete crack modeling

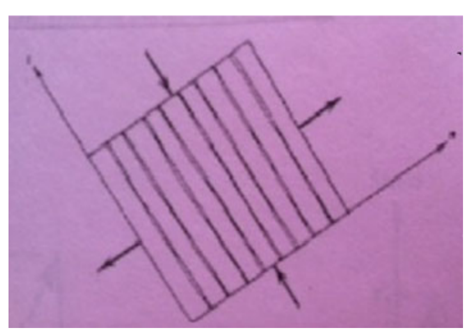


Fig. 4 Smeared crack modeling

crack approach it is assumed that initially isotropic material becomes isotropic at the onset of cracking with the principal axes of material oriented along the direction of crack and does not change throughout the analysis. In contrast to permanent memory of the fixed crack approach the rotating crack approach assumes crack orientation continuously changing and assumes the "current most active crack". In this approach, the initially isotropic material remains isotropic and any change in material properties at a point happens in all direction, however, in fixed crack approach, there was no control over the maximum tensile strength in other direction at the same gauss point and therefore over estimates the strength of the structure.

A concrete-based model for the consideration of tension stiffening has been used in the present study to simulate the post-cracking behavior of concrete. For reinforcement, the constitutive relations have been formulated by defining yield criterion with an isotropic hardening rule and associative flow rule. The Von Mises criterion describes the yielding of steel in the present study.

Material Properties

Isotropic material properties	
Young's modulus EX	3.36×10^{10} N/m^2 (Specimen 1)
Young's modulus EX	2.83×10^{10} N/m^2 (Specimen 2)
Young's modulus EX	3.22×10^{10} N/m^2 (Specimen 3)
Young's modulus EX	3.28×10^{10} N/m^2 (Specimen 4)
Density DENS	25,000 N/m^3
Poisson's ratio NUXY	0.20

EX is the modulus of elasticity of the concrete (E_c), and NUXY is the Poisson's ratio (υ). The modulus of elasticity of the concrete has been evaluated from the equation $E_c = 5{,}000\sqrt{f_c}$ where fc is characteristic compressive strength of concrete in N/mm^2.

Poisson's ratio for the concrete has been assumed to be 0.20. The compressive uniaxial stress–strain relationship for the concrete model has been obtained using the following equations to compute the multi linear isotropic stress–strain curve for the concrete [9].

$$f = (E_c\varepsilon)/\left(1 + (\varepsilon/\varepsilon_o)^2\right)$$

$$\varepsilon_o = (2f'c \;/\; E_c)$$

$$E_C = (f \;/\; \varepsilon)$$

where f is the stress at any strain ε and ε_o is the strain at the ultimate compressive strength f_c

The uniaxial crushing stress in this model is based on the uniaxial unconfined compressive strength (f' c) and is denoted as f_t. This has been not considered in the analysis due to convergence problem, as suggested by the scientists [8].

The Link8 element has been used for all the steel reinforcement in the beam and it is assumed to be bilinear isotropic. EX is the modulus of elasticity of the steel (E_c), and the PRXY is the poisson's ratio (υ).

These have been taken as 2.0×10^{11} N/m^2 and 0.35 respectively.

After putting the material properties, finite element analysis software itself takes care of bond between concrete and reinforcement bar.

Results and Discussion

The effect of detailing of steel on joints under opening bending moment has been studied with help of load deflection curves. The load versus deflection curves for corners Specimen 1, Specimen 2, Specimen 3 and Specimen 4 obtained numerically using finite element analysis software. The load deflection plots for all the specimens as obtained by finite element analysis software have been compared with corresponding experimental plots as obtained by the earlier researches [1]. This comparison has been shown in Fig. 5.

Load Deflection Behavior of Specimen 1 (Detailing System 1)

It can be seen from Fig. 5 that up to 3.2 mm deflection experimental results almost coincides with results obtained

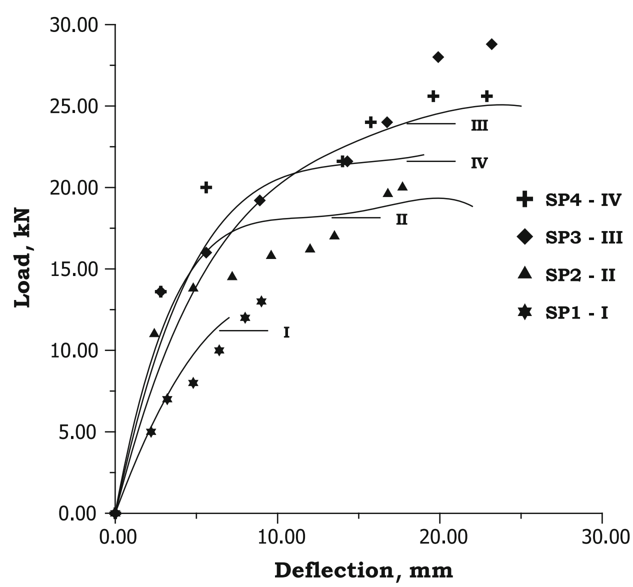


Fig. 5 Comparison between analytical results and experimental results

from finite element analysis software. Experimental specimen took 12 kN load and 7 mm deflection at ultimate failure and corresponding load and deflection for finite element analysis software model has been observed to be 13 kN and 9 mm respectively.

Load Deflection Behavior of Specimen 2 (Detailing System 2)

Until the appearance of first crack i.e. up to elastic limit finite element analysis software results coincides with experimental results. After the elastic region finite element analysis software model takes lesser load for the same value of deflection. After 13.5 mm deflection finite element analysis software model shows higher values for load as compared to experimental one. At ultimate failure experimental model takes 22 mm deflection and 19 kN load but finite element analysis software model takes 17.7 mm deflection and 20 kN load.

Load Deflection Behavior of Specimen 3 (Detailing System 3)

In elastic region up to 8.9 mm deflection results of both almost coincides with each other. At ultimate failure finite element analysis software results are on little bit higher side for load i.e. 23.2 mm deflection and 28.8 kN load instead of 25 mm deflection and 25 kN load for experimental specimen.

Load Deflection Behavior of Specimen 4 (Detailing System 4)

For elastic region finite element analysis software model gives little bit higher values of load for same value of deflection. At ultimate failure deflection and load for finite element analysis software model are 22.9 mm and 25.6 kN respectively and for experimental model these values are 17 mm and 22 kN respectively.

Percentage error in load and deflection are calculated and has been presented in Table 2. It can be seen from Table 1 that the small difference exists between finite element analysis software and experimental results. This can be due to following reasons:

1. As finite element analysis software consider ideal conditions while giving results where as in experiments variations in strengths for constituent's members may exist.
2. Moreover while modeling in finite element analysis software, no cover was provided to reinforcement.

Table 1 Analytical and experimental comparison between deflection and load

	Deflection, mm		Load, kN	
	Analytical	Experimental	Analytical	Experimental
SP1	9	7	13	12
SP2	17.7	18.4	20	19
SP3	23.2	25	28.8	25
SP4	22.9	24	25.6	22

Table 2 Percentage error in load and deflection

Models	% age error in load	% age error in deflection
Specimen 1	8.33	28.5
Specimen 2	5.26	3.8
Specimen 3	15.2	7.2
Specimen 4	16.3	4.58

Table 3 Ductility Index for models

S. no.	Model no.	Deflection, mm	Ductility Index $(d_2 - d_1)/d_1$
1	Specimen 1	$d_2 = 9$ mm $d_1 = 3.625$ mm	1.483
2	Specimen 2	$d_2 = 17.7$ mm $d_1 = 5.0$ mm	2.54
3	Specimen 3	$d_2 = 23.2$ mm $d_1 = 6.0$ mm	2.867
4	Specimen 4	$d_2 = 22.9$ mm $d_1 = 6.0$ mm	2.8167

Ductility Behaviour

The ductility behaviour of these models in flexure has also been investigated. The ductility index has been defined as:

Ductility Index = (d2 − d1)/d1, where d_1 and d_2 are the deflection corresponding to first crack load and ultimate load.

The values of Ductility Index for all the models are presented in Table 3. It can be seen from Table 3, that Specimen 3 comes out to be best. But From the practical point of view, in corners with narrow dimensions, it becomes difficult to bend the rebar in the form of a loop and place the reinforcement cage in the proper position. This problem is further aggravated if the section is heavily reinforced or if larger diameter bars are used or if there is a significant difference in the dimensions of the two members framing into the corner. Detailing system 4 used in Specimen 4 is essentially a compromise of the loop used in detailing system 3 and it affords easier fabrication of the rebar cage. Detailing system 4 promises to some extent the confining action of the loop by filling out the corner while at the same time, offering considerable ease in fabrication.

Specimen 4 comes out to be best among all the four models made in finite element analysis software, so the failure mechanism and parametric study is done on Specimen 4.

The results are as given below:

Failure Mechanism for Specimen 4 (Figs. 6, 7, 8, 9):

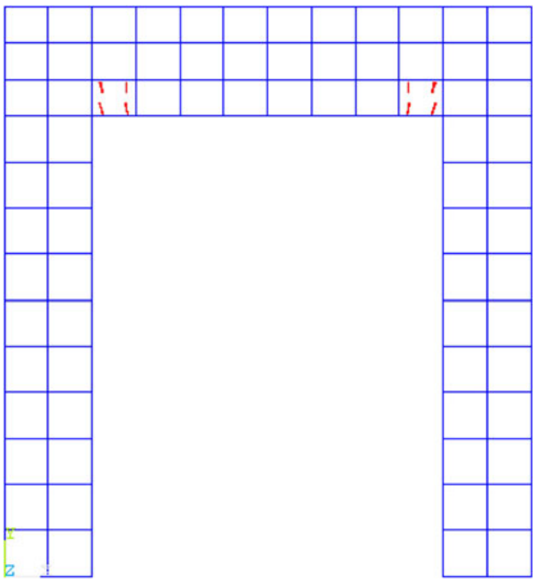


Fig. 6 Crack pattern for the Specimen 4 (finite element analysis software ANSYS) (At 12 kN load)

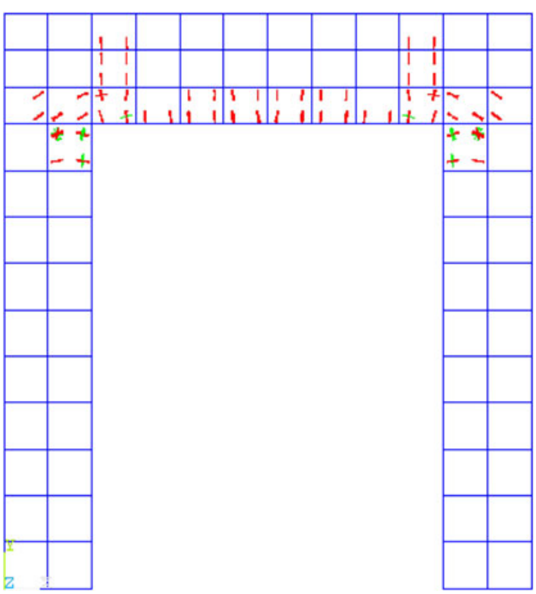


Fig. 7 Crack pattern for the Specimen 4 (finite element analysis software ANSYS) (At 20.2 kN)

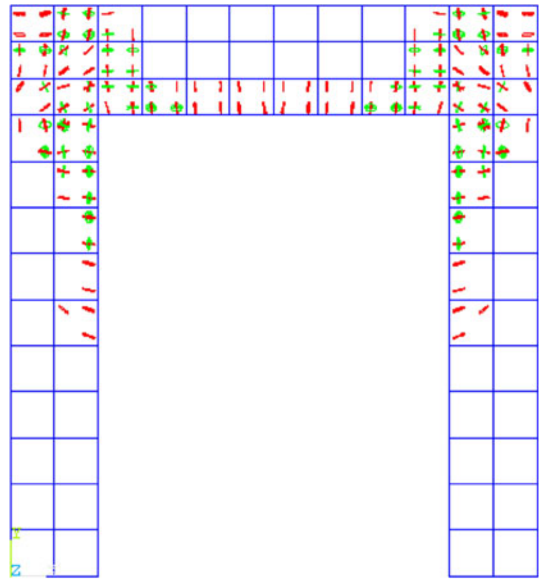


Fig. 8 Crack pattern for the Specimen 4 (finite element analysis software ANSYS) (At 25.2 kN)

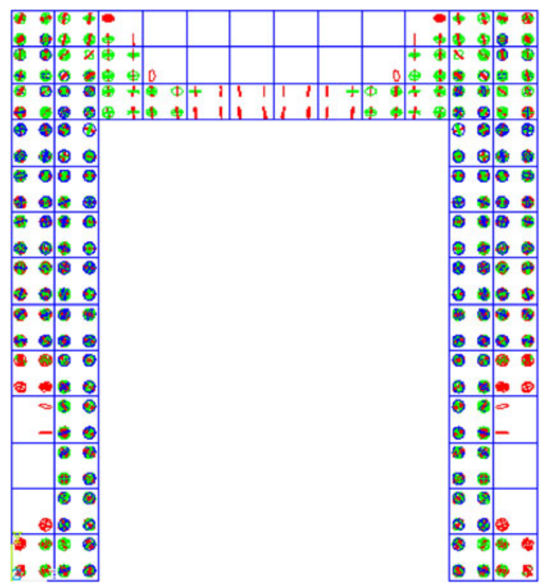


Fig. 9 Crack pattern for the Specimen 4 (finite element analysis software ANSYS) (At ultimate failure)

Small vertical, horizontal and inclined lines represent the initial cracks and small circles represent the widened cracks.

The first crack appear in the inside portion of horizontal member near the two corners. Load and deflection at the first crack was 12 kN and 2.5 mm respectively. In the 10th sub step cracks propagate in horizontal member upward. Cracks can also be seen up to half of diagonal portion of corner. Cracks appear in whole the horizontal member near the location of 12 mm diameter bars and start appearing in vertical members as shown in Fig. 8. Load and deflection was 20.2 kN and 6.25 mm respectively. At 30th sub step

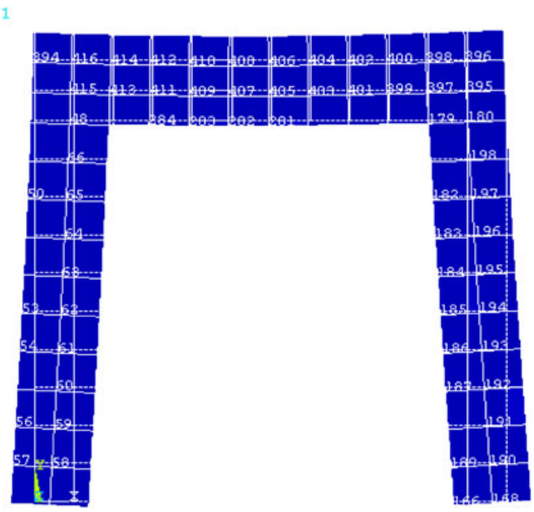


Fig. 10 Deformed shape for Specimen 4 (finite element analysis software ANSYS)

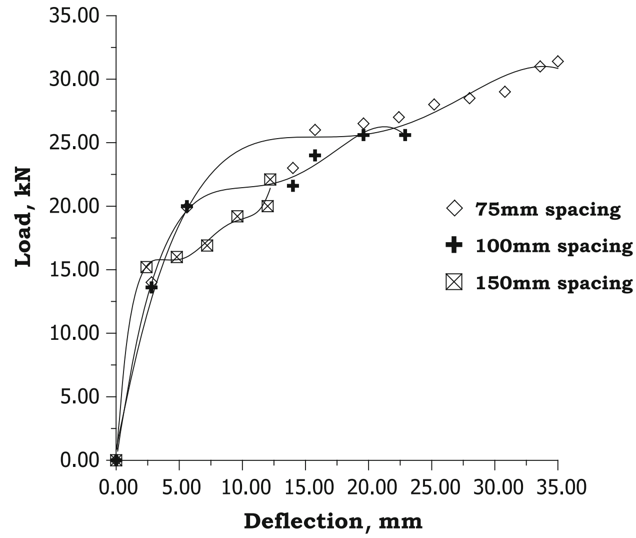


Fig. 11 Load deflection curves for Specimen 4 with variable spacing of shear reinforcement

some cracks appear in outer portion of vertical members also. Cracks increase downward in inner side of both vertical members. Load and deflection was 25.2 kN and 18.75 mm respectively. At ultimate failure more cracks appear in outer portion of vertical members. In inner portion of vertical members cracks appear up to full length. Load and deflection at ultimate failure was 25.6 kN and 22.9 mm respectively.

Deformed shape for Specimen 4 has been shown in Fig. 10 at ultimate failure.

As the results obtained from finite element analysis software were comparable with experimental results, this gave confidence in the use of finite element analysis software and the models developed. The approach was then utilized to analyze the models for parametric study.

Parametric Study

As Specimen 4 comes out to be best among all the four models made in finite element analysis software, so parametric study is done on Specimen 4. In this, three parameters are studied. First parameter is spacing of shear reinforcement, second parameter is grade of concrete and third parameter is diameter of main steel.

By keeping all other parameters same spacing of shear reinforcement is varied in Specimen 4 (Fig. 11).

1. Shear reinforcement with spacing 75 mm c/c. The only change is in values of real constants for concrete. Real constants for model

Real constants for rebar 1	
Material number	4
Volume ratio	0.00670
Orientation angle (θ1)	90°
Orientation angle (Ø1)	0
Real constants for rebar 2	
Material number	5
Volume ratio	0.00383
Orientation angle (θ2)	0
Orientation angle (Ø2)	90°

The volume ratio can be calculated as below:

$$\frac{\left[\left(\frac{\Pi}{4}\right)\times\left(\frac{8}{1000}\right)^2\times(0.200+0.200)\right]}{[0.200\times0.350\times0.075]}=0.00383$$

$$\frac{\left[\left(\frac{\Pi}{4}\right)\times\left(\frac{8}{1000}\right)^2\times(0.350+0.350)\right]}{[0.200\times0.350\times0.075]}=0.00670$$

At the appearance of first crack load and deflection is 20.9 kN and 8.33 mm respectively. As the load is increased, deflection also increases and at ultimate failure load and deflection taken by this model are 31.4 kN and 35 mm respectively.

2. Shear reinforcement with spacing 100 mm c/c. The only change is in values of real constants for concrete. The volume ratio can be calculated as below:

$$\frac{\left[\left(\frac{\Pi}{4}\right)\times\left(\frac{8}{1000}\right)^2\times(0.200+0.200)\right]}{[0.200\times0.350\times0.100]}=0.00287$$

$$\frac{\left[\left(\frac{\Pi}{4}\right)\times\left(\frac{8}{1000}\right)^2\times(0.350+0.350)\right]}{[0.200\times0.350\times0.100]}=0.00502$$

At the appearance of first crack load and deflection is 20.01 kN and 5.67 mm respectively. As the load is increased, deflection also increases and at ultimate failure load and deflection taken by this model is 25.6 kN and 22.9 mm respectively.

3. Shear reinforcement with spacing 150 mm c/c. The only change is in values of real constants for concrete. The volume ratio can be calculated as below:

$$\frac{\left[\left(\frac{\Pi}{4}\right)\times\left(\frac{8}{1000}\right)^2\times(0.200+0.200)\right]}{[0.200\times0.350\times0.150]}=0.00191$$

$$\frac{\left[\left(\frac{\Pi}{4}\right)\times\left(\frac{8}{1000}\right)^2\times(0.350+0.350)\right]}{[0.200\times0.350\times0.150]}=0.00335$$

At the appearance of first crack load and deflection is 15.4 kN and 3 mm respectively. As the load is increased, deflection also increases and at ultimate failure load and deflection taken by this model is 22.1 kN and 12.2 mm respectively.

Ductility for Models with Variable Spacing of Shear Reinforcement

The ductility behaviour of these models in flexure has also been investigated. Ductility values for all the models are given in Table 4.

After comparing all the curves as given in Fig. 11 and ductility values as given in Table 4, it is concluded that model Specimen 4 having shear reinforcement of spacing 75 mm *c/c* gives the best results because it takes the maximum load with maximum deflection or it has highest ductility among the three. Gradually on increasing the spacing the ductility goes on decreasing.

Table 4 Ductility Index for models with variable spacing of shear reinforcement

S. no.	Specimen 4	Deflection, mm	Ductility Index $(d_2 - d_1)/d_1$
1	With 150 mm c/c spacing	$d_2 = 12$ mm $d_1 = 3$ mm	3
2	With 100 mm c/c spacing	$d_2 = 22.9$ mm $d_1 = 5.67$ mm	3.03
3	With 75 mm c/c spacing	$d_2 = 35$ mm $d_1 = 8.33$ mm	3.20

Load versus Deflection behaviors of Specimen 4 with the change in Grade of concrete are given in Fig. 12. In this the change is made in values of material properties for concrete only.

With Concrete of Grade M-20 Young's modulus $EX = 2.236 \times 10^{10}$ N/m^2.

At the appearance of first crack load and deflection is 9.52 kN and 1.904 mm respectively. As the load is increased, deflection also increases and at ultimate failure load and deflection taken by this model is 12 kN and 8 mm respectively

With Concrete of Grade M-25 Young's modulus $EX = 2.50 \times 10^{10}$ N/m^2.

At the appearance of first crack load and deflection is 10.66 kN and 1.904 mm respectively. As the load is increased, deflection also increases and at ultimate failure load and deflection taken by this model is 12.91 kN and 8.5 mm respectively

With Concrete of Grade M-30 Young's modulus $EX = 2.739 \times 10^{10}$ N/m^2.

At the appearance of first crack load and deflection is 8.50 kN and 3.21 mm respectively. As the load is increased, deflection also increases and at ultimate failure load and deflection taken by this model is 14.4 kN and 16.2 mm respectively.

With Concrete of Grade M-35 Young's modulus $EX = 2.96 \times 10^{10}$ N/m^2.

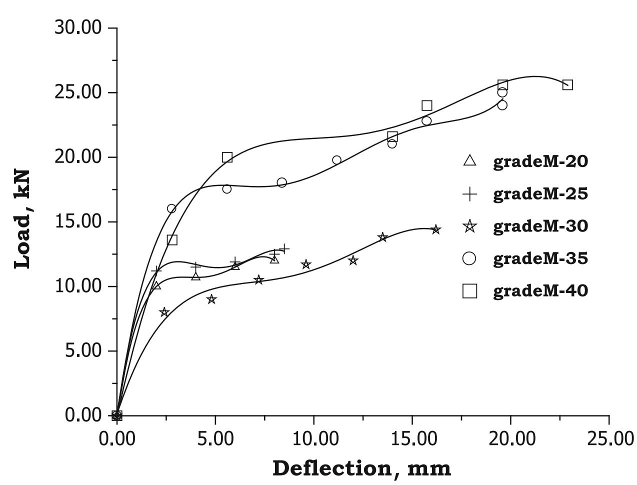


Fig. 12 Load deflection curves for Specimen 4 with variable grades of concrete

At the appearance of first crack load and deflection is 16.41 kN and 3.57 mm respectively. As the load is increased, deflection also increases and at ultimate failure load and deflection taken by this model is 25 kN and 19.6 mm respectively.

With Concrete of Grade M-40 Young's modulus EX = 3.28×10^{10} N/m^2.

At the appearance of first crack load and deflection is 16.8 kN and 4.2 mm respectively. As the load is increased, deflection also increases and at ultimate failure load and deflection taken by this model is 25.6 kN and 22.9 mm respectively.

Ductility Index for Models Having Variable Concrete Grades

The ductility behaviour of these models in flexure has also been investigated. Ductility Index for all the models are given in Table 5. From values given in Table 5, it is concluded that model Specimen 4 having concrete Grade M-40 gives the best result because it takes the maximum load with maximum deflection and having maximum ductility value. Gradually on decreasing the grades the ductility goes on decreasing.

By keeping all the parameters same diameter of main bars is varied in Specimen 4 (Fig. 13).

1. Only 12 mm diameter bars are used
2. Along with 12 mm diameter bars 16 mm diameter bars are also used
3. Only 16 mm diameter bars are used

The ductility behavior of these models in flexure has also been investigated. Ductility values for all the models are given in Tables 2, 3 and 5.

After comparing all the curves from Fig. 13 and ductility values as given in Table 6, it is concluded that model Specimen 4 having diameter of main bars as 12 mm diameter only gives the best result because it takes the maximum load with maximum deflection and having maximum ductility value.

Table 5 Ductility Index for models having variable concrete grades

S. no.	Specimen 4	Deflection, mm	Ductility Index $(d_2 - d_1)/d_1$
1	Concrete Grade 20	$d_2 = 8$ mm $d_1 = 1.904$ mm	3.2
2	Concrete Grade 25	$d_2 = 8.5$ mm $d_1 = 1.904$ mm	3.46
3	Concrete Grade 30	$d_2 = 16.2$ mm $d_1 = 3.21$ mm	4.04
4	Concrete Grade 35	$d_2 = 19.6$ mm $d_1 = 3.57$ mm	4.49
5	Concrete Grade 40	$d_2 = 22.9$ mm $d_1 = 4.2$ mm	4.5

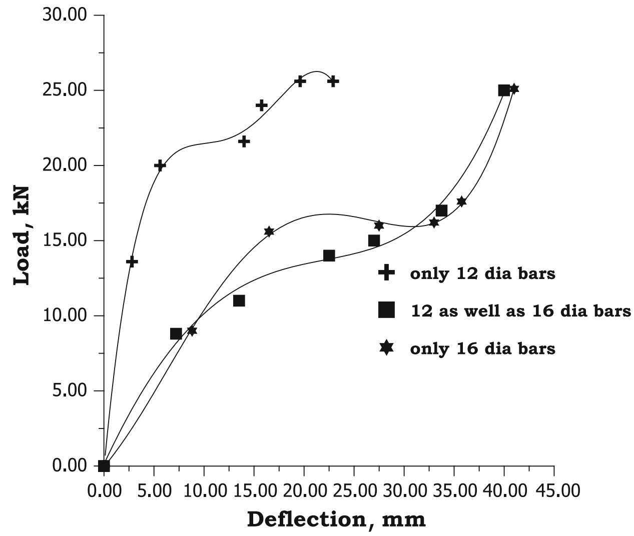


Fig. 13 Load deflection curves for Specimen 4 with variable dia of main steel

Table 6 Ductility Index for models having variable dia of main steel

S. no.	Specimen 4	Deflection, mm	Ductility Index $(d_2 - d_1)/d_1$
1	Only 12 mm diameter bars	$d_2 = 22.6$ mm $d_1 = 2.5$ mm	8.04
2	Along with 12 mm diameter bars 16 mm diameter bars	$d_2 = 40$ mm $d_1 = 7.5$ mm	4.33
3	Only 16 mm diameter bars	$d_2 = 41$ mm $d_1 = 8.75$ mm	3.68

Conclusions

The main conclusions drawn are summarized below:

The analytical results show good agreement with the experimental results. Although the ultimate load of Specimen 3 is highest among the four models but its fabrication is very difficult and not practical especially in corners with narrow dimensions. Among all the models, Specimen 4 comes out with best results because it affords easier fabrication of the rebar cage and at the same time its load at ultimate failure and its ductility value almost matches with Specimen 3.

It is concluded that model Specimen 4 having concrete Grade M-40 gives the best result because it takes the maximum load with maximum deflection and having maximum ductility value.

It is concluded that model Specimen 4 having shear reinforcement of spacing 75 mm c/c gives the best results because it takes the maximum load with maximum deflection or it has highest ductility among the three. Gradually on increasing the spacing the ductility goes on decreasing.

It is concluded that Specimen 4 having main dia of 12 mm gives the best results as it takes maximum load with maximum deflection and having maximum ductility value.

References

1. B. Singh, S.K. Kaushik, Investigations on fibre reinforced concrete opening corners. J Inst Eng (India) **84**, 201–209 (2003)
2. A.R.C. Murthy, G.S. Palani, N.R. Iyer, T.V.S.R. Appa Rao, An efficient FE modelling strategy for fracture analysis of tubular joints. J Inst Eng (India) **85**, 17–25 (2004)
3. V.N. Dhar, P.K. Singh, Chamfering and reinforcement detailing in reinforced concrete corner subjected to opening moment. J Inst Eng (India) **84**, 244–251 (2004)
4. W.F. Chen, Y.M. Lan, *Finite element study of confined concrete*, Proceedings of International Symposium on Confined Concrete, Changsha, China 21–34 (2004)
5. W.F. Chen, *Plasticity in reinforced concrete* (McGraw-Hill, New York, 1982)
6. W.F. Chen, A.F. Saleeb, *Constitutive equations for engineering materials, Vol. 1: elasticity and modeling* (Wiley, New York, 1982)
7. A. Mirmiran, K. Zagers, W. Yuan, Nonlinear finite element modeling of concrete confined by fiber composites. Finite Elem Anal Des **35**(1), 79–96 (2000)
8. B.M. Kavlicoglu, F. Gordaninejad, M. Saiidi, Y. Jiang, *Analysis and testing of graphite/epoxy concrete bridge girders under static loading*, Proceedings of Conference on Retro-fit and Repair of Bridges, London, England, (2001)
9. J.G. MacGregor, *Reinforced concrete mechanism and design* (Prentice-Hall Inc., Englewood Cliffs, 1992)