# INVESTIGATION OF THE SWAY CHARACTERISTICS OF REINFORCED CONCRETE FRAMES SUBJECTED TO LATERAL LOADS

A thesis submitted by


**Muhammad Badre Enam**


In partial fulfillment of the requirements for the degree of

**Master of Science in Civil Engineering**




Department of Civil Engineering

Bangladesh University of Engineering & Technology


August, 2001

# INVESTIGATION OF THE SWAY CHARACTERISTICS OF REINFORCED CONCRETE FRAMES SUBJECTED TO LATERAL LOADS

A thesis by

**Muhammad Badre Enam**

Approved as to the style and content by the Examination Committee on 6th August 2001.


**Dr. Khan Mahmud Amanat** — Chairman (Supervisor)
Associate Professor
Department of Civil Engineering
BUET, Dhaka.

**Dr. Md. Abdur Rouf** — Member
Professor and Head
Department of Civil Engineering
BUET, Dhaka.

**Dr. Alamgir Habib** — Member
Professor
Department of Civil Engineering
BUET, Dhaka.

**Dr. Ishtiaque Ahmed** — Member
Associate Professor
Department of Civil Engineering
BUET, Dhaka.

**Dr. Sk. Reaz Ahmed** — Member (External)
Assistant Professor
Department of Mechanical Engg.
BUET, Dhaka

# DECLARATION

Declared that, except where specific references are made to other investigators, the work embodied in this thesis is the result of investigation carried out by the author under the supervision of Dr. Khan Mahmud Amanat, Associate Professor, Department of Civil Engineering, BUET.

Neither the thesis nor any part thereof is submitted or is being concurrently submitted in candidature for any degree at any other institution.

Muhammad Badre Enam

Author

# TABLE OF CONTENETS

**Chapter 4: DEFORMATION CAHARACTERISTICS OF RC JOINTS**



**Chapter 5: STUDY OF THE SWAY BEHAVIOUR OF RC FRAME**

**Chapter 6: A RATIONALE FOR SWAY ANALYSIS**



**Chapter 7: CONCLUSION**

# ACKNOWLEDGEMENTS

The author wishes to convey his profound gratitude to almighty Allah for allowing him to bring this effort to fruition. The initial notion of this thesis was conceived by Dr. Khan Mahmud Amanat, Associate Professor, Department of Civil Engineering, BUET. His systematic guidance and constant persuasion has immensely helped the author throughout this work. The author expresses his indebtedness his superior technical expertise, impromptu answers and solutions to numerous computer-related problems.

Also the author pays his deepest homage to his parents, whom he believes to be the cardinal source of inspiration for all his achievements. Their constant support throughout this work was phenomenal and exemplary. Also the author is appreciative of his brother and sister for their unabated assistance.

Last but not the least, the author commends his wife for her moral support through out this work.

# ABSTRACT

Sway of reinforced concrete (RC) frames subjected to lateral load is an important design parameter from serviceability point of view. A reasonably accurate determination of sway still lags proper guideline. The widely followed code of American Concrete Institute (ACI) as well as Bangladesh National Building Code (BNBC) does not provide any specific guideline in this regard. These codes only suggest the use of effective moment of inertia in plane frame idealization of RC frames using linear frame members and homogenous material properties. Such approximation of RC frames may be considered as over simplification by present day standards with the availability of versatile computational techniques. Thus there is scope of investigation and improvement of the methods for sway estimation.

In this thesis an investigation based on finite element (FE) modeling of RC frame is carried out to study the sway behaviour of RC frame. In the FE modeling the heterogeneity of RC is maintained by separate modeling of concrete, steel and bond-slip mechanism using different elements. Thus the RC frame is modeled as two-dimensional plane stress problem in which bending as well as shear deformations are included. Flexural cracks are also incorporated. Thus developing a finite element model that is closer to reality, an extensive investigation is performed under various parametric conditions to determine the deformation and sway characteristics of RC frames.

The investigation is based on two models. The first model consists of two columns and a beam representing an exterior joint of a RC frame. Investigation of deformation characteristics of this frame by FE modeling as well as ACI guideline established the fact that ACI method differs from the FE analysis in terms of deflections calculated. The second FE model was a simple portal frame subjected to a lateral load. The same frame was analyzed according to ACI guideline and the results are compared with those of FE analysis. From the comparison, different parameters, which influence the sway, are identified. It was observed that in most of the cases ACI method tends to overestimate the sway by considerable amount, establishing the fact that there is scope of modification in the ACI guideline.

In the last phase of the work investigation was carried out to identify possible areas of modification in the ACI formula and proposals have been made for calculation of effective moment of inertia using which more reasonable estimation of sway can be made. Finally the performance of the proposals are checked through an example.

Chapter 1

# INTRODUCTION

## 1.1 GENERAL

Lateral deflection or sway of frame structures is an important design parameter from serviceability point of view. This sway property can turn out to be a governing factor in member proportioning of large Reinforced Concrete (RC) buildings. As a general rule the deflection of a frame structure should be a consideration in selecting the sections of its members. Generally the smaller the member the larger is the deflection or vice versa. Therefore to reduce the sway and in turn to reduce member stresses, sections are to be increased. This leads to a rise in cost of the structure and renders the structure uneconomical. Determination of sway of RC frames subjected to lateral loads by conventional methods tends to overestimate the sway values to remain on the safe side. Sometimes such estimation becomes unnecessarily high resulting in uneconomical sections of members that are, in terms of strength, more than adequate. Efforts are therefore required to avoid this situation and proper tools need to be designed which would help to estimate the sway of RC structures more accurately under lateral loads. With the help of such tools, reasonably accurate determinations of the lateral deflections of RC buildings would be possible in design offices which would ultimately contribute to eliminate wastage of material in designing the various structural members of the building.

## 1.2 LATERAL DEFLECTION OF RC FRAMES

With increasing use of high strength concrete and steel, frame members are getting thinner resulting in higher amount of deflection. In order to keep deflection or sway within permissible limit defined by various codes of practices, designers are often forced to choose larger sections for frame members nullifying the benefits of using high strength material. American Concrete Institute (ACI) is one of the leading authorities in the world in formulating design practices for RC construction. The guidelines laid out by ACI are widely practiced throughout the world. Prior to 1993, ACI code was virtually the only design code followed in the design and construction of RC structures in Bangladesh. Bangladesh National Building Code (BNBC), formulated in 1993, also gives guidelines for RC construction. However the BNBC specifications are not much different from those of ACI and ACI still remains to be more elaborate and comprehensive compared to BNBC. Apart from providing design criteria for various concrete constructions ACI also recommends analytical procedures for RC structures. Following this code one can calculate the design moments and shears of a structure under various loading conditions and hence design various members with adequate factor of safety. However the ACI code does not provide us with any specific recommendation regarding sway estimation of frame structures. Since most of the modern day RC building constructions consists of beam and column arrangements, the most elementary structural form of these buildings would be a simple portal frame. The ACI code proposes methods to calculate the member forces of a portal frame. But it does not recommend any analytical procedure for the calculation of lateral deflection. In fact the only deflection estimation method it proposes is only for simply supported and continuous beams.

In the absence of any guideline regarding sway analysis of RC structures a reasonable method need to be devised to overcome this problem. One tool can be employed in this purpose which has emerged in its full glory during the past two decades. This technique is popularly known as **Finite Element** method. With the advent of powerful desktop computers this technique has gained immense popularity in design offices. Using finite element method it is possible to model any RC structure to its most minute details. Analysis can then be performed on this model for a better solution of its sway characteristics. However this method has its shortcomings too. Finite element modeling is not meant for day to day analysis of structures and the modeling itself requires some degree of expertise. Also it becomes a huge task if one has to model the minute details of high-rise RC building of considerable magnitude.

Requirements of a more reasonable approach are therefore evident. The approach should be simpler to implement and should be based on extensive analysis to ensure its reliability in the design office. Here in this thesis attempts will be made to propose one such technique that would fulfill these criteria.

## 1.3 SCOPE AND OBJECTIVE

As discussed earlier the most basic structural form of a RC beam and column structure is a simple portal frame. Hence the approach described in this thesis would emphasize on the sway characteristics of a simple RC portal structure. Following the deductions from its analysis a more general-purpose proposition would be made which would be applicable for any arrangements of a number of portal frames. However, to start with the sway analysis of the portal frame, one factor must be considered which is more important a RC structure than for structures made of different materials. This is the factor of joint continuity of the beam and column connection. Conventional frame analysis has some limitations such as a) shear deformations are neglected in the analysis and b) the frame members are assumed homogenous and there is no scope of having any cracked sections. Conventional analysis and design of RC frame structures are usually carried out under the assumption that the connections joining the beams to the columns are fully rigid. This rigid joint assumption implies that full slope continuity exists between the adjoining members and that the full (or a substantial percentage) of gravity moment is transferred from the beam to the column. Although the assumption of fully rigid connection behaviour drastically simplifies the analysis and design procedure, the validity of the assumption is questionable in light of the fact that concrete, due to its high capacity in compression and presence of reinforcement with its inherent tensile and ductile characteristics may impart to the connection some degree of non-rigidity.

In the light of the above argument attempts would be made to analyze the joint behaviour of a beam and column connection. The approach would be to carry out a finite element modeling of the joint with arrangements for some degree of semi-rigidity within the joint. Establishing the fact that, there does exists some rotational behaviour in the connections, the same methodology would be incorporated in the analysis of RC portal frame.

During the next phase of analysis i.e. during the analysis of the portal structure, calculation of the sway characteristics of the structure would be performed under lateral loads. The deflection would be calculated for various geometrical and component parameters of the structure and would be compared with those found from conventional analysis in light of the ACI guidelines. Based on this comparison attempts would be made to propose some modifications of the conventional analysis, which would be applicable to RC structures. Finally these modifications would be applied to conventional analysis of different structures with different portal frame arrangements. Side by side finite element analysis of those structures would also be carried out to check the performance of the proposed modifications of the conventional technique.

## 1.4 ASSUMPTIONS AND CONTENTS

The investigation described in this thesis assumes a linear behaviour of all the material properties. Also the finite element modeling was limited to two-dimensional approach.

The whole thesis is organized into seven chapters. Chapter 1 is the current chapter, which introduces the work presented in this thesis. Chapter 2 concentrates on the deflection analysis of RC structures. Chapter 3 introduces the finite element modeling of reinforced concrete structures. Joint rotation behavior is focussed in Chapter 4. Chapter 5 deals with the finite element analysis of the portal frame structure and compares the results with that of the results obtained by ACI deflection calculation method. Depending on the differences of these two sets of results a new rationale is proposed in Chapter 6. The performance of the proposed rationale is also focussed in Chapter 6. And finally, Chapter 7 draws conclusions by summarizing the outcome of the thesis and proposes new direction for further research and development.

Chapter 2

# DEFLECTION OF REINFORCED CONCRETE FRAME

## 2.1 INTRODUCTION

In order to analyze the deflections of frame structure one needs to get familiarized with the techniques available to this date. Since the current study concentrates mainly on the sway of a RC frame structure, discussion would be made about the methods for deflection calculation as proposed in the ACI manual. ACl code does not state any direct technique for the estimation of sway of a RC frame. It mainly manifests the methods of deflection calculation of simply supported and continuous RC beams. In the absence of any guideline towards the sway analysis for frames, the technique proposed for continuous beams would be adopted and attempts would be made to derive an acceptable solution to the problem. Therefore, here in this chapter, details of the deflection estimation techniques of beams as described in the ACI committee report ACI 435R-95 is introduced. Finally the features and limitations of this approach would be highlighted at the end of this chapter.

## 2.2 BEHAVIOUR OF RC FRAMES UNDER LATERAL LOADS

Since any study should start with an analysis of the current state of the art report about the subject matter, efforts would now be made to investigate the achievements regarding sway of RC frame structures by other researchers. Actually the amount of research work carried out on lateral deflection of RC frames appears to be very sparse, if not rare. On the other hand, considerable amount of work has been done regarding the sway of steel frames.

Since 1930, there has been considerable research into the behaviour of steel structural connections and hence sway of steel structures. A number of investigations have been carried out to measure the moment-rotation characteristics of various type of steel framing connections. Baker (1931, 1935) and Rathbun (1936) used slope deflection and moment distribution methods to analyze frames with semi-rigid connections. Batho and Rowan (1934) presented a beam line method for analyzing semi-rigid frames. Monforton and Wu (1963) incorporated the effects of connection deformations into a stiffness analysis program. However, they assumed linear moment-rotation characteristics. Sommer (1969) developed a procedure for expressing the moment-rotation characteristics for all connections of a given type, in a standardized, nondimensional form. More recent works in this field include building of connection databases from experimental tests by Goverdhan (1983), Nethercot (1985a, 1985b) and Kishi and Chen (1986).

A number of researchers have worked on RC frames or RC joint behaviour subjected to lateral loads. Alameddine and Ehsani (1991) have done extensive investigation on high-strength RC connections subjected to inelastic cyclic loading. The primary variables for their test specimens were concrete compressive strength, joint shear stress and joint transverse reinforcement. Their study ultimately revealed that in spite of the brittle nature of plain high-strength concrete, properly detailed frames constructed with high-strength concrete exhibit ductile hysteric response similar to those for ordinary-strength concrete.

Sheikh, Deierlein, Yura and Jirsa (1989) had carried out another study on beam and column moment connections for composite frames. Although their study involves investigation and design of steel members, the work is mentioned here because concrete also plays an important part in composite joints. The above researchers studied the behaviour of composite beam and column connections through results of an experimental research program where 15 two-thirds scale joint specimens were tested under monotonic and cyclic loading. The study revealed that composite beam and column connections are reliable details that provide adequate stiffness at service loads and ductile failure at ultimate loads. Additionally, under reversed cyclic loading, composite connections exhibited toughness comparable to reinforced concrete joints detailed for seismic design.

Qi and Pantazopoulou (1991) studied the experimental response of a single-story, indeterminate RC frame with floor slabs. The frame was quarter-scale and tested under static lateral-load reversals simulating earthquake load. The internal force redistribution occurring within the inelastic range of response and the beam flexural overstrength resulting from slab contributions were evaluated. Specimen behaviour was characterized in terms of strength, stiffness, lateral drift ratio and plastic hinge formation. It was observed that the pattern of lateral- load distribution in indeterminate structures is severely affected by the partial restraint that continuity imposes on the inelastic member expansion, particularly at large levels of lateral displacement.

Mehrabi, Shing, Schuller and Noland (1997) performed a comprehensive research, more relevant to the current study. They studied the influence of masonry infill panels on the seismic performance of reinforced concrete frames. Two types of frames were considered. One was designed for wind loads and the other for strong earthquake forces. Twelve ½ -scale, single-story, single-bay frame specimens were tested. The parameters investigated included the strength of infill panels with respect to that of the bounding frame, the panel aspect ratio, the distribution of vertical loads and the lateral load history. The experimental results indicated that the infill panels could significantly improve the performance of RC frames. However, specimens with strong frames and strong panels exhibited a better performance than those with weak frames and weak panels in terms of the load resistance and energy dissipation capability. The lateral loads developed by the infilled frame specimens were always higher than that of the bare frame.

It is evident, therefore, that practically no work on estimation of sway of RC frames under lateral load is inadequate. As discussed in chapter 1, sway values have large influence on beam and column member sections and hence dictates the economy in RC design. Also the sway of a structure cannot be allowed to increase indefinitely. There are guidelines in all codes of practices regarding the maximum allowable sway of a structure. ACI recommends that the sway of an RC structure should not exceed H/480 or 0.002% of H, where H is the total height of the structure. Therefore, it is evident that there is scope of investigating methods of predicting sway of RC frames and hence come up with some guideline, which may enable us to predict the sway of RC frame structures with reasonable amount of accuracy.

## 2.3 ASSUMPTIONS OF ACI TECHNIQUE

The ACI method of deflection calculation is based on the idea of *Effective moment of inertia* $I_e$. Tension cracks occur at sections when the imposed loads cause bending moments in excess of the cracking moment. Cracks develop at several sections along the member length. While the cracked moment of inertia, $I_{cr}$, applies to the cracked sections, the gross moment of inertia, $I_g$, applies to the uncracked concrete between these sections. According to the extensive studies by Branson (1977, 1982, and 1985), an effective moment of inertia of the section can be evaluated using combinations of the moment of inertia of cracked sections and the moment of inertia of gross section. This effective moment of inertia $I_e$ then can be used to calculate the deflection once the maximum actual moment of the section has exceeded the cracking moment of the section.

## 2.4 REQUIRED DEFINITIONS

To fully understand the ACI method of deflection calculations some definitions need to be discussed.

***Concrete modulus of rupture,*** $f_r$**:** ACI 318 (1999) recommends the following equation for computing the modulus of rupture of concrete with different densities,

$$f_r = 7.5\lambda\sqrt{f_c'}\text{ , psi} \qquad 2.1$$

where $f_c'$ = Compressive strength of concrete

$\lambda$ = 1.0for normal density concrete [145 to 150 pcf]
= 0.85 for semi low-density concrete [110-145 pcf]
= 0.75 for low-density concrete [90 to 110 pcf]

***Gross moment of inertia, $I_g$:*** This is the second moment of area of the cross section about the neutral axis. Since our study is limited to beam and columns of rectangular cross section the gross moment of inertia is given by

$$I_g = \frac{bh^3}{12}$$

where $b$ and $h$ are dimensions as shown in fig. 2.1.

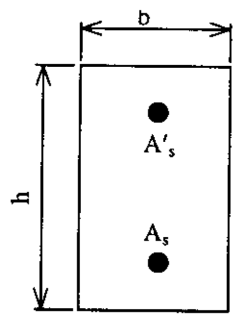



**Fig. 2.1: Section of doubly reinforced beam**

***Cracking moment of inertia, $I_{cr}$:*** This is considered the moment of inertia of the section after it has suffered cracking. This is most likely to happen when a concrete member is subjected to bending. The cracking moment of inertia hence neglects the portion of the cross section, which has cracked and therefore is always less than the gross moment of inertia $I_g$. The calculation of $I_{cr}$ for a rectangular, doubly reinforced beam or column section is as follows.

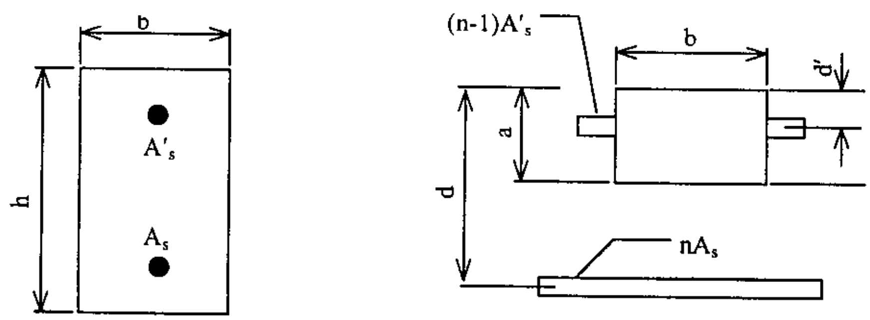



**Fig 2.2: Cracked transformed section**

$$B = \frac{b}{nA_s} \qquad r = \frac{(n-1)A_s'}{nA_s} \qquad a = \frac{\sqrt{2db\left(1+\frac{rd'}{d}\right)+(1+r)^2}-(1+r)}{B}$$

$$I_{cr} = \frac{ba^3}{3} + nA_s(d-a)^2 + (n-1)A_s'(a-d')^2 \qquad 2.2$$

***Cracking moment, $M_{cr}$:*** This is section property and is calculated as follows,

$$M_{cr} = \frac{f_r I_g}{y_t} \qquad 2.3$$

where $f_r$ = Modulus of rupture

$I_g$ = Gross moment of inertia

$y_t$ = Distance from the neutral axis to the tension face of the beam.

## 2.5 ESTIMATION OF EFFECTIVE MOMENT OF INERTIA

The following discusses the calculation of effective moment of inertia, $I_e$. It is to be noted that Branson proposed this method of estimating $I_e$. This approach was selected as being sufficiently accurate to control deflections in reinforced and prestressed concrete structural elements like beams.

***Simply supported beams:*** Branson's equation for the effective moment of inertia $I_e$, for short-term deflections is as follows,

$$I_e = \left(\frac{M_{cr}}{M_a}\right)^3 I_g + \left[1-\left(\frac{M_{cr}}{M_a}\right)^3\right] I_{cr} \le I_g \qquad 2.4$$

where $M_a$ = Maximum service load moment (unfactored) at the stage for which deflections are being considered

***Continuous beams:*** For continuous members, ACI 318-99 stipulates that $I_e$ may be taken as the average value obtained from the same equation as the one for simply supported beams for the critical positive and negative moment sections. For prismatic members, $I_e$ may be taken as the value obtained at midspan for continuous beams. The use of midspan section properties for continuous prismatic members is considered satisfactory in approximate calculations primarily because the midspan rigidity including the effect of cracking has the dominant effect on deflections (ACI 435, 1978).

If one chooses to average the effective moment of inertia $I_e$, then according to ACI 318-99, the following expression should be used,

$$I_e = 0.5I_{e(m)} + 0.25\left(I_{e(1)} + I_{e(2)}\right) \tag{2.5}$$

where the subscripts m, 1 and 2 refer to midspan, and two beam ends, respectively.

Improved results for continuous prismatic members can, however, be obtained using a weighted average as presented in the following equations:

For beams continuous on both ends,

$$I_e = 0.70I_{e(m)} + 0.15\left(I_{e(1)} + I_{e(2)}\right) \tag{2.6}$$

For beams continuous on one end only,

$$I_e = 0.85I_{e(m)} + 0.15\left(I_{e(1)}\right) \tag{2.7}$$

## 2.6 DEFLECTION CALCULATION

Once $I_e$ is calculated using equation 2.4 and if necessary equation 2.5, 2.6 and 2.7, the deflection can be obtained using the moment-area method. For the current study a scheme was used which involves repetitive solutions of the structure using stiffness method.

## 2.7 REMARKS ON EFFECTIVE MOMENT OF INERTIA

A close observation of the equations required for $I_e$ estimation reveals the following facts.

- $I_e$ provides a transition between the upper and the lower bounds of $I_g$ and $I_{cr}$, respectively, as a function of the level of cracking, expressed as $M_{cr}/M_a$. Equation 2.4 clearly shows that $I_e$ is an interpolation between the well defined limits of $I_g$ and $I_{cr}$. Hence it can be safely written that,

$$I_{cr} < I_e < I_g \qquad 2.8$$

- When the value of $M_a$ is zero the term $M_{cr}/M_a$ would return infinite value. This would in turn make the value of $I_e$ infinity, which is unacceptable. In this case $I_e$ should equal the value $I_g$. This may be explained in the light of the fact that when there is no moment within a section, there is no possibility of cracks. And due to the absence of cracks the whole cross-section should be considered for calculating $I_e$.
- From the detailed calculation process of $I_g$, $I_{cr}$ and $I_e$, as described above, it can be conferred that heavily reinforced members will have an $I_e$ approximately equal to $I_{cr}$, which may in some cases (flanged members) be larger than $I_g$ of the concrete section alone.
- For most practical cases, the calculated $I_e$ will be less than $I_g$ and should be taken as such in the design for deflection control, unless a justification can be made for rigorous transformed section computations.

## 2.8 LIMITATIONS

From the study of equation 2.4 it is evident that for the determination of $I_e$, the actual moment of the section $M_a$ is required. Now $M_a$ can only be obtained by analyzing the structure after loads have been applied on it. This poses a dilemma where the value of $M_a$ has to be known beforehand to calculate the sway using $I_e$. To overcome this problem one approach can be used. At the very beginning of the analysis the members should have moment of inertia equal to their gross moment of inertia i.e. $I_g$. With this assumption the structure can be analyzed with full application of load. Upon the completion of analysis the values for actual moment $M_a$ can be obtained.

With these values of $M_a$ the effective moment of inertia, $I_e$ of all the sections can be calculated. Using the values of $I_e$ obtained it would be possible to perform deflection calculations.

However, this process would produce another problem. The second time calculation for the deflection incorporating $I_e$ would in this case give values of $M_a$ that would be different from the values obtained from the first analysis. Now to stop the calculation at this level and to take the results of the second analysis as the correct ones would be somewhat questionable. This problem hence was overcome in the study using iteration process. Thus the values of $M_a$ from the second analysis were taken and once again $I_e$ was calculated. With this new $I_e$ another solution was obtained and corresponding values of $M_a$ was calculated. These new values of $M_a$ were utilized again to obtain new $I_e$. This process was then continued for a number of times until the values of $M_a$ obtained from two consecutive analyses were close enough. At this stage the $I_e$ was calculated for the last time using the results from the last analysis and deflection was calculated accordingly. The whole process can be summarized using the following flowchart.

## 2.9 REMARKS

The effective moment of inertia method outlined in ACI committee report 435R-95 has been proposed mainly for simply supported and continuous beams. However it can be equally applied for continuous beam-column arrangements of any combination. Future chapters would hence analyze a simple portal frame using this method.

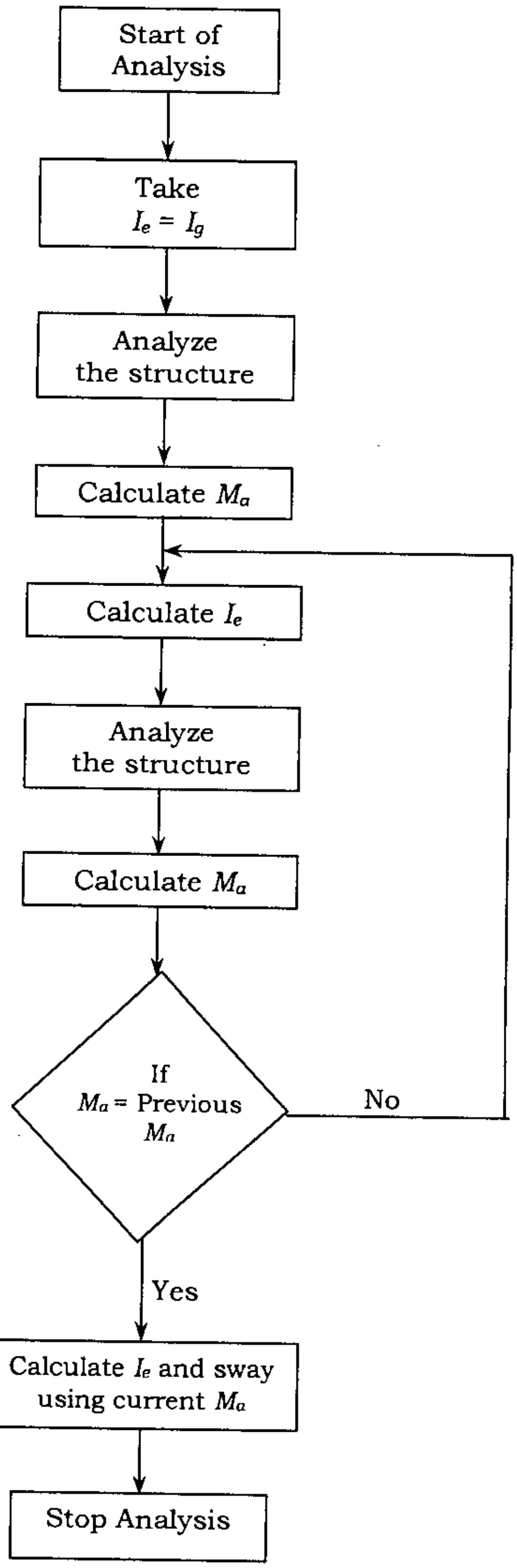



**Fig. 2.3: Flowchart for ACI effective moment of inertia method**

## Chapter 3

# FINITE ELEMENT MODELING OF REINFORCED CONCRETE

## 3.1 INTRODUCTION

The actual work regarding the finite element modeling of reinforced concrete joint has been described in detail in this chapter. Representation of various physical elements with the FEM (Finite Element Modeling) elements, properties assigned to them, representation of various physical phenomenon such as bond slip, reinforcement behaviour etc. have also been discussed.

## 3.2 THE FINITE ELEMENT PACKAGE

A number of good finite element analysis computer packages are available in the market. They vary in degree of complexity, usability and versatility. The names of such packages are

- Micro Feap
- SAP 90
- FEMSKI
- ANSYS
- ABAQUS
- MARC
- ADINA
- DIANA

Of these the package *ANSYS* used in this study for its relative ease of use, detailed documentation, flexibility and vastness of its capabilities. The version of *ANSYS* used was *Version 5.4*.

*ANSYS* Finite Element analysis software enables engineers to perform the following tasks.

- Build computer models or CAD models of structures, products, components and systems.
- Apply operating loads and other design performance conditions.
- Study the physical responses, such as stress levels, temperature distributions, or the impact of electromagnetic fields.
- Optimize a design early in the development process to reduce production costs.
- Do prototype testing in environments where it otherwise would be undesirable or impossible (for example, biomedical applications).

The *ANSYS* program has a comprehensive graphical user interface (GUI) that gives user easy, interactive access to program functions, commands, documentation and reference material. An intuitive menu system helps users navigate through the *ANSYS* program. Users can input data using a mouse, a keyboard, or a combination of both.

## 3.3 FINITE ELEMENT MODELING OF RC

Reinforced concrete, speaking in a very common sense, is a mass of hardened concrete with steel reinforcement embedded within it. This arrangement when in use acts as a single material with the steel providing adequate tensile capacity to concrete which has high capacity to take compression. However the interaction between the concrete mass and the steel reinforcement is not a very simple one when subjected to various loading conditions. Complicated physical phenomenon such as bond slip, anchorage etc. comes into play in this condition. Hence the whole of reinforced concrete may not be treated as a single material during FEM analysis and may not be modeled as a unique composite material.

The modeling of reinforced concrete, as outlined in this thesis, used separate materials and elements for the concrete and steel reinforcement. Although a special composite concrete element with provisions for reinforcement could have been used, it was not used in the modeling. The separate treatment in the element level ensures better approximation of the actual condition. Also to simulate the bond slip phenomenon to its full extent special combination elements were provided at the interface between the concrete and the steel reinforcement.

### 3.3.1 Modeling of Concrete

Since the whole modeling was confined to two-dimensional analysis all the elements used were 2-D in nature. For representing the concrete two types of 2-D quadrilateral elements were used. Owing to simplicity and due to the inherent physical characteristics of the model all the elements produced with these two base elements were assumed as rectangular planes. The following is a discussion of the two base elements used.

**4 Node Quadrilateral Element:** This type of element is generally used for 2-D modeling of solid structures. The element can be used either as a plane element or as an axisymmetric element. The element is defined by four nodes having two degrees of freedoms at each node: translations in the nodal x and y directions. The element has plasticity, creep, swelling, stress stiffening, large deflection, and large strain capabilities.

**Input data:** The element input data includes four nodes, a thickness (for plain stress option only) and the orthotropic material properties. Orthotropic material directions correspond to the element coordinate directions.

**Output data:** The solution output associated with the element is in the form of nodal displacements included in the overall nodal solutions. The element stress directions are parallel to the element coordinate.

**Assumptions and restrictions:** The area of the element must be non-zero. The element must lie in a global X-Y plane and the Y-axis must be the axis of symmetry for axisymmetric analyses. An axisymmetric structure must be modeled in the +X quadrants.

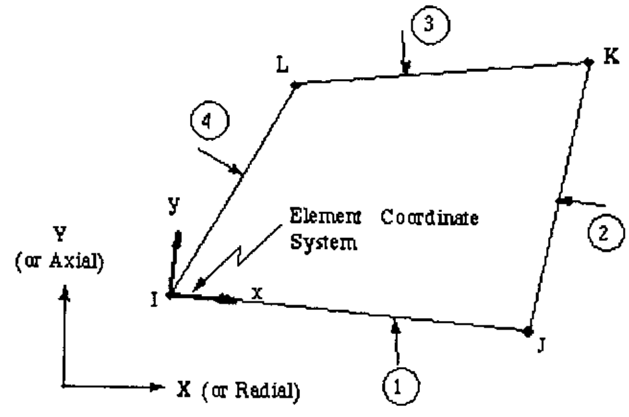



**Fig 3.1: 4 Node quadrilateral element**

**8 Node Quadrilateral Element :** This element is a higher order version of the two dimensional four node element. It provides more accurate result for mixed (quadrilateral-triangular) automatic meshes and can tolerate irregular shapes without much loss of accuracy. The 8-node elements have compatible displacement shapes and are well suited to model curved boundaries.

The 8-node elements defined by eight nodes having two degrees of freedom at each node: translations in the nodal x and y directions. The element may be used as plane element or as an axisymmetric element. The element has plasticity, creep swelling, stress stiffening, large deflection, and large strain capabilities.

**Input data:** Element input data include node locations, thickness (for plane stress option only) and the orthotropic material properties. Orthotropic material directions correspond to the element coordinate directions. Midside nodes may be removed (with a zero node number) to form a pattern compatible with other element types. The geometric locations of midside nodes are automatically calculated, if not supplied.

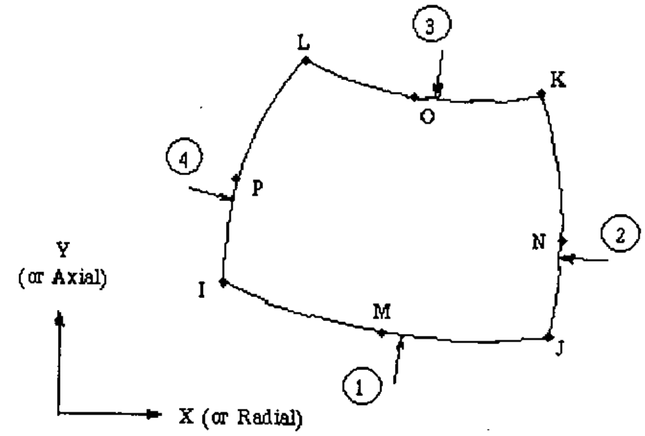



**Fig 3.2: 8 Node quadrilateral element**

**Output data:** The solution output associated with the element is in the form of nodal displacements included in the overall nodal solution. The element stress directions are parallel to the element coordinate system. Surface stresses are defined parallel and perpendicular to the IJ face (and the KL face) and along Z- axis for a plane analysis or in the hoop direction for an axisymmetric analysis.

**Assumptions and restrictions:** The area of the element must be positive. The element must lie in the global X-Y plane. The Y-axis must be the axis of symmetry for axisymmetric analyses. An axisymmetric structure should be modeled in the +X quadrants. A face with a removed midside node implies that the displacements vary linearly, rather than parabolically, along the face.

## 3.3.2 Modeling of Reinforcement

To simulate the reinforcements both in beam and column one-dimensional link elements were used. Only one type of link element was used, the details of which have been described below.

**2 Node Link Element:** This element can be used in variety of engineering applications. Depending upon the application, it can be thought as a truss element, a link, a spring etc. The two-dimensional spar element is a uniaxial tension-compression element with two degrees of freedom at each node: translations in the nodal x and y directions. As in a pin-jointed structure, no bending of the element is considered.

**Input data:** Two nodes, the cross-sectional area, an initial strain and the material properties define the element. The element x-axis is oriented along the length of the element from node I toward node J. The initial strain in the element is given by $\delta/L$, where $\delta$ is the difference between the element length, L, and the zero -strain length.

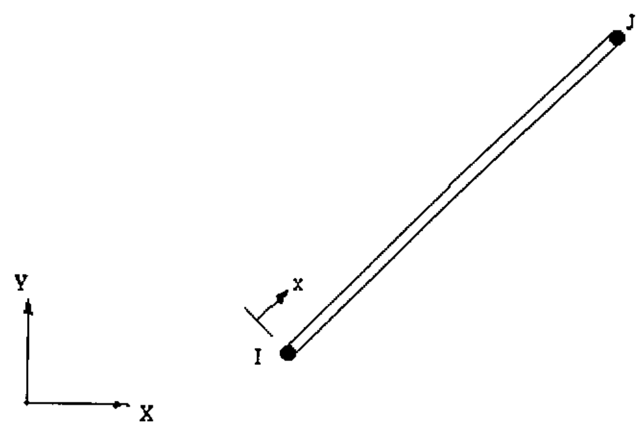



**Fig. 3.3: 2 Node link element**

**Output data:** The solution output associated with the element is in the form of nodal displacements included in the overall nodal solution.

**Assumptions and restrictions:** The spar element assumes a straight bar, axially loaded at its ends, of uniform properties from end to end. The length of the spar must be greater than zero. Therefore nodes J and I must not be coincident. The spar must lie in an X-Y plane and must have an area greater than zero. The displacement function implies a uniform stress in the spar. The initial strain is also used in calculating the stress stiffness matrix, if any, for the first cumulative iteration.

### 3.3.3 Modeling of Bond-slip

One of the most remarkable features of the Finite Element Modeling was the simulation of bond slip that actually exists between the concrete and steel in any reinforced concrete system. The element that was used to simulate the bond slip characteristics in the Finite Element Model will be discussed a little later. But first discussion would be carried out on the bond-slip property itself.

**Bond stress-slip relationship:** Bond behaviour is a combination of adhesion, bearing of lugs and friction. Adhesion is related to the shear strength of the steel-concrete interface, which is basically a result of the chemical bonding. Bearing forces perpendicular to the lug faces arise as the bar is loaded and tries to slide. In this phase microcracking and micro crushing of concrete at the front of the lugs are produced. Friction is produced by the bearing force on the interactional surface and by shearing of the concrete between the lugs on the cylindrical concrete surface at the tip of the lugs. It has been well established (CEB 1996) that bond stress is a basic function of slip where the latter indicates the relative displacement between the steel and concrete cross section. This relationship is called bond stress-slip relationship.

For analytical applications several linear and non-linear approximations of the bond stress-slip relationship are available. Dorr (1980) proposed a non-linear function relating the bond stress with the tensile strength and relative slip as

$$f_b = f_t\left[5(\delta/\delta_0) - 4.5(\delta/\delta_0)^2 + 1.4(\delta/\delta_0)^3\right] \quad , \quad 0 < \delta < \delta_0$$

where $f_b$ = nominal bond stress

$f_t$ = tensile strength of concrete in MPa

$\delta$ = local bond slip

$\delta_0$ = amount of slip at which perfect slip occurs, usually taken as 0.06 mm.

In the present analysis, however, a simplified linear form of the formula proposed by Amanat (1997) has been used. This is shown in the following figure.

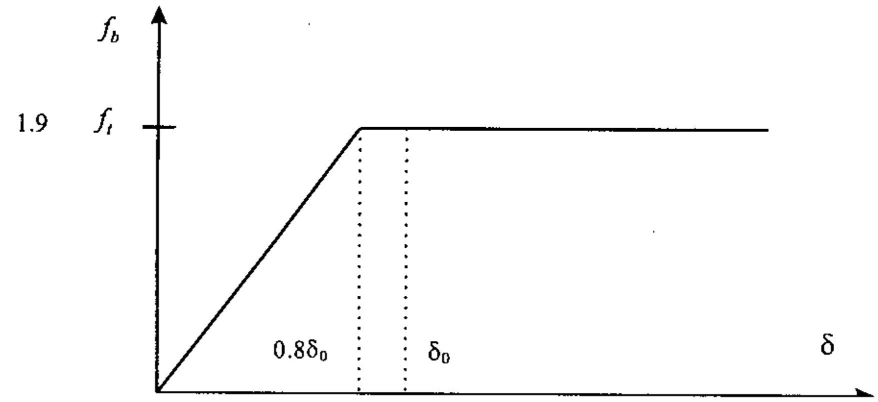



**Fig. 3.4: Simplified bond stress-slip relation**

For the modeling described in this thesis the value of $f_b$ has been taken as constant at $1.9f_t$. This is a simple approximation following our assumption of $\delta / \delta_0 = 1$.

$$f_b = f_t[5 - 4.5 + 1.4] = 1.9f_t$$

The bond stress-slip relationship is simulated by the use of the spring elements. These contact elements are dimensionless bond link elements, connecting a single concrete node to a corresponding reinforcement node.

This bond element is basically a simple spring, in our case a spring damper element in-between two co-incident nodes of 2-node link element and 4-node quad element or those of 2-node link and 8-node quad element. This spring is in the direction of the reinforcement (parallel) which simulates the bond stress-slip relationship. The spring constant may be evaluated as follows.

Total surface area = $\pi DLn$ ,

where $D =$ bar dia.

$L =$ effective length of bar

$n =$ number of bars

Total force = $\pi DLn \times 1.9f_t = 1.9\pi DLnf_t$

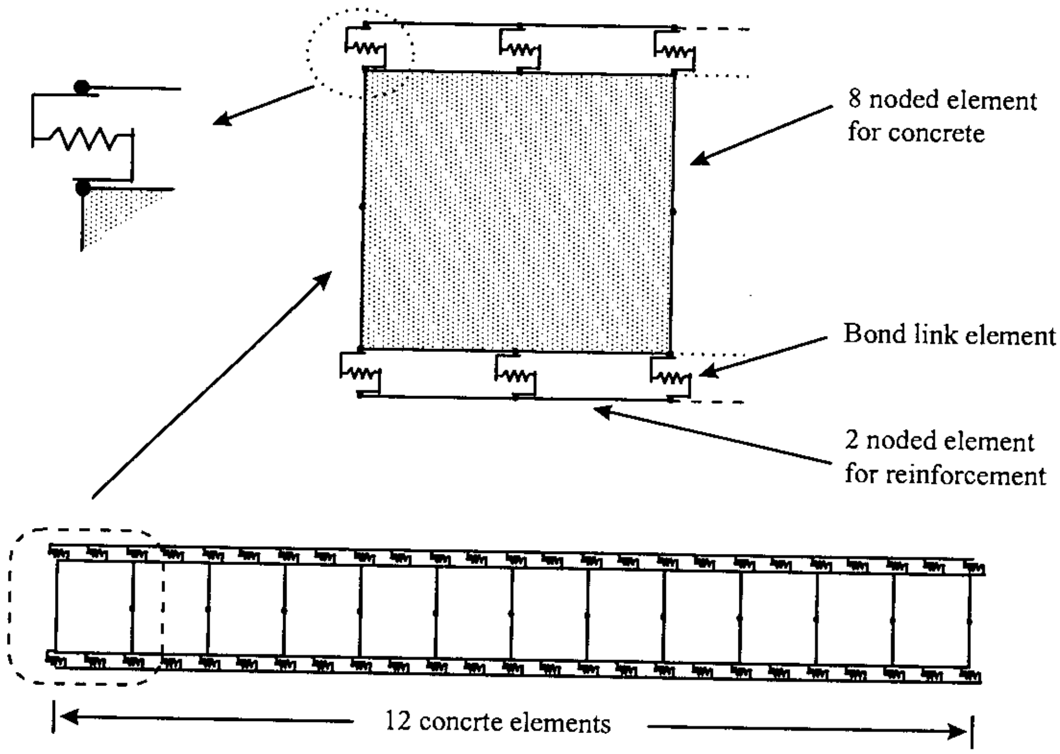



**Fig. 3.5: FEM modeling of bond stress-slip relationship**

Amount of slip for occurrence of perfect slip $\delta_0$ = 0.06 mm. $= 2.3622 \times 10^{-3}$ in.

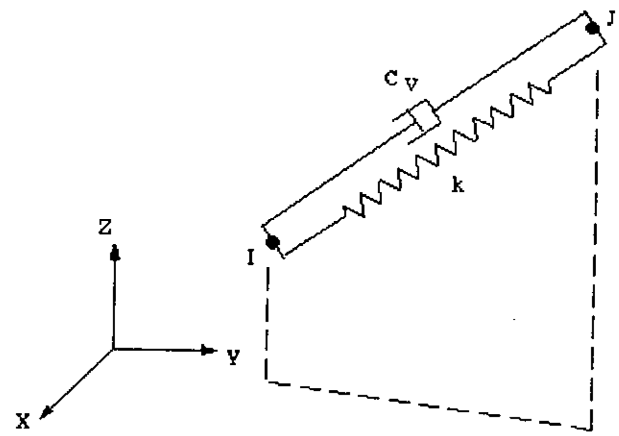
$$\text{Stiffness } k = \frac{F}{\delta}$$

$$= \frac{1.9\pi DLnf_t}{2.3622 \times 10^{-3}}$$

$$= 2526.888DLnf_t$$

Having discussed the estimation of bond-slip property focus is set on the element that is used to simulate this phenomenon in the finite element modeling.

**Spring Element:** This element has longitudinal and torsional rigidity in one, two, or three-dimensional applications. The longitudinal spring-damper option is a uniaxial tension-compression element with up to three degrees of freedom at each node: translations in the nodal x, y and z directions. No bending or torsion is considered. The torsional spring-damper element is a purely rotational element with three degrees of freedom at each node: rotations about the nodal x, y and z axes. No bending or axial loads are considered. The spring-damper element has no mass.




Note - Two-dimensional elements must lie in the X-Y plane

**Fig. 3.6: Spring element**

**Input data:** The element is defined by two nodes, a spring constant k and damping coefficients $cv_1$ and $cv_2$. The damping capability is not used for static or undamped modal analyses. The longitudinal spring constant should have units of Force/Length, the damping coefficient units are Force×Time/Length. The torsional spring constant and damping coefficient have units of Force×Length/Radian and Force×Length×Time/Radian respectively.

**Output data:** The solution output associated with the element is in the form of nodal displacements included in the overall nodal solution.

**Assumptions and restrictions:** The longitudinal spring element stiffness acts only along its length. The torsion spring element stiffness acts only about its length, as in a torsion bar. The element allows only a uniform stress in the spring. The spring or damping capability may be deleted from the element by setting k or cv equal to zero.

## 3.4 REMARKS

This chapter describes the essentials of finite element modeling of reinforced concrete. From this point on all the discretization of RC structural members would be followed using the guidelines outlined in this chapter. However in some point forward it may be required to introduce special arrangements within the modeling. The discussion of such extra features would be discussed where appropriate.

Chapter 4

# DEFORMATION CAHARCTERSITICS OF RC JOINTS

## 4.1 INTRODUCTION

Sway of a framed structure subjected to lateral load is controlled by two principal factors - a) member stiffness and b) joint characteristics. Both factors are well addressed in the analysis of steel frames. In RC frames, however, it is customary to consider the member stiffness only assuming that the joints are rigid. This usually leads to some degree of approximation in the sway estimation of RC frames. A correct and rational prediction of sway behaviour of RC framed structures requires the knowledge of semi-rigid characteristics of joints. In a RC framed structure, beam to column joints are perhaps among the most complicated yet one of the least understood components of a building system. Proper understanding of the joint characteristics is one of the most challenging fields among researchers. It has long been recognized that the key to appreciating the effects of joint performance on the behaviour of frames is the knowledge of the connection's moment-rotation (M-$\phi$) characteristics. The ACI committee report 352R-91 suggests that *the designer should consider the possible effect of joint rotations on cracking and deflection.* This however is not backed by ACI with any guideline whatsoever. Therefore, conventional analysis and design of RC frame structures are usually carried out under the assumption that the connections joining the beams to the columns are fully rigid, which implies that full slope continuity exists between adjoining members and that the full (or a substantial percentage) of gravity moment is transferred from the beam to the column. This assumption drastically simplifies the analysis and design of RC structures. However it should be kept in mind that unlike steel, reinforced concrete is not a homogenous material. The principal component, concrete, can withstand large amount of compression with very poor performance in tension. The other component, reinforcement, on the other hand, has prominent tensile and ductile characteristics. The combination of these two materials, hence, is likely to behave as a semi-rigid material. And if that is the case there is a good possibility of rotational characteristics within a beam and column joint made of RC. Here in this chapter the aim is to investigate whether such rotational behaviour really exist in a joint. If such behaviour is evident attempts would be made to work out the difference between the results obtained from two sets of analyses – one, considering full slope continuity of joints; and two, incorporating semi-rigid characteristics within the connection.

## 4.2 STUDY OF A TYPICAL RC JOINT

As discussed in the section above, analysis of a RC structure would be carried out using two techniques with special emphasis on joint rotation characteristics. For the first analytical technique the effective moment of inertia method as proposed in ACI 435R-95 would be adopted. This method has been discussed in detail in chapter 2. It is worth mentioning that this method does not identify the semi-rigid behaviour of RC and hence assumes full slope continuity of beams to columns. For the second analysis finite element method would be used. The details of discretization of reinforced concrete using finite elements have been discussed in the previous chapter. Using finite element technique it is possible to incorporate bond-slip mechanism at the interface of concrete and reinforcement, which seems primarily responsible for the joint's semi-rigid behaviour.

Having decided on the techniques of analyses, the next step would be to choose a frame structure upon which the above methods would be applied. Obviously the structure to be studied should consist of at least a single joint and some arrangements of beams and columns. For the current investigation an exterior joint with beam and column system has been chosen as presented in the skeletal structure in fig 4.1.

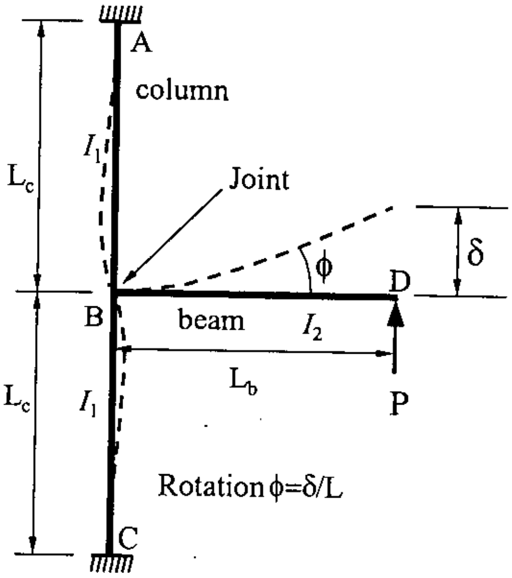



**Fig. 4.1 Exterior joint with beam and column arrangement**

It is evident that the RC connection has a beam and two columns framing into it. The beam and each of the columns has the same length of 10 feet. Other specifications such as beam depth, amount of reinforcement etc. has been varied for a limited parametric study of the structure. As seen in the skeletal structure the column ends are fixed while the beam end away from the joint is considered free.

## 4.3 ANALYSIS METHODOLOGY

The objective of the current study is to identify the impact of joint semi-rigidity on the deflection pattern of the structure shown in fig. 4.1. In this regard the structure would be subjected to a certain pattern of loading for both types of analysis and then the associated deflection characteristics would be compared. As shown in the skeletal structure the free end of the cantilever is subjected to a certain amount of load *P*, which in this case is 5 kips. Due to this loading the deflection of the free end of the cantilever, $\delta$ is measured. The whole process is repeated for certain variations of the section properties of the beams and columns. Finally all the deflection values are tabulated for both conventional analysis and finite element analysis. This data is then utilized for the parametric study of the structure. Conclusion is then made on the rotational behaviour of the RC joint following the results of the parametric study.

## 4.4 CONVENTIONAL ANALYSIS USING ACI TECHNIQUE

Guidelines as represented in the ACI committee report 435R-95 was used for this set of analyses. The effective moment of inertia $I_e$ was calculated for both beams and columns. The deflection of the free end was then calculated using stiffness method. Due to the repetition involved spreadsheet analysis package was used for this purpose.

## 4.5 FINITE ELEMENT ANALYSIS

Finite element modeling of the RC structure was carried out using the discretization technique described in chapter 3. However some prominent features of the modeling requires elaboration.

### 4.5.1 Critical Section

A beam column joint should be proportioned to resist forces at the critical sections. The critical sections for transfer of member forces to the joint are at the joint-member interfaces. Design recommendations are based on the assumption that the critical sections are immediately adjacent to the joint. Exceptions, however, are made for joint shear and reinforcement anchorage. These specifications are written in ACI committee report 352R-91. Current investigation has tried to follow its recommendations as closely as possible. Hence calculation of moment at the connection was carried out at the interface of the beam and the joint.

## 4.5.2 Effective Depth Consideration

At the bottom of a beam framing into a joint, when the tension stress $f_{ct}$ exceeds modulus of rupture cracks form as shown in fig. 4.2. If the concrete compression stress is less than approximately $0.5f_c'$ and the steel stress has not reached the yield point, both materials continue to behave elastically, or very nearly so. In this situation, which is generally obtained in structures under normal service conditions and loads, for simplicity and with little error if any, it is assumed that tension cracks have progressed all the way to the neutral axis and plane sections before bending are plane in the bent member.

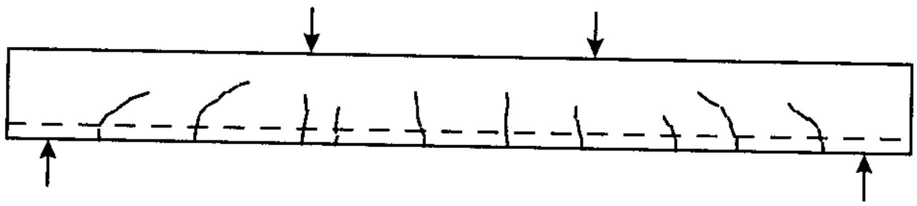


**Fig. 4.2: Cracking of a beam under load**

For simplicity in computation, the fact that all the concrete, which is stressed in tension, is assumed cracked and therefore effectively absent, can be taken into consideration. In this situation a transformed section is obtained that consists of the concrete in compression on one side of the axis and *n* times the steel area on the other side. Once the concrete is cracked, any material located below the steel is ineffective and hence the idea of effective depth *d* evolves.

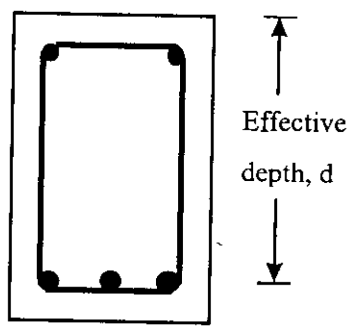



**Fig. 4.3: Effective depth of beam**

Following the deduction, it can be concluded that from the viewpoint of stress-strain and moment calculation, inclusion of concrete below the steel may be redundant. Hence current modeling of the beam part framing into the joint has only a depth equal to the effective depth *d*.

### 4.5.3 Aspect Ratio

An important characteristic of the FE modeling that affects an analysis is the aspect ratio of the plane elements. The aspect ratio describes the shape of the element in the assemblage. For two-dimensional elements, this parameter is conveniently defined as

$$\text{Aspect Ratio} = (\text{Largest dimension} \div \text{Smallest dimension})$$

The optimum aspect ratio at any location within the grid depends, largely upon the difference in rate of change of displacements in different directions. If the displacements vary at about the same rate in each direction, the closer the aspect ratio to unity, the better the quality of solution. In other words, efforts are made avoid long narrow elements. The effects of various aspect ratios on the representativeness of the modeling have been illustrated in the next page.

Within the scope of the finite element modeling of the exterior joint discussed in this text, special care was taken to maintain an aspect ratio close to unity for all the elements.

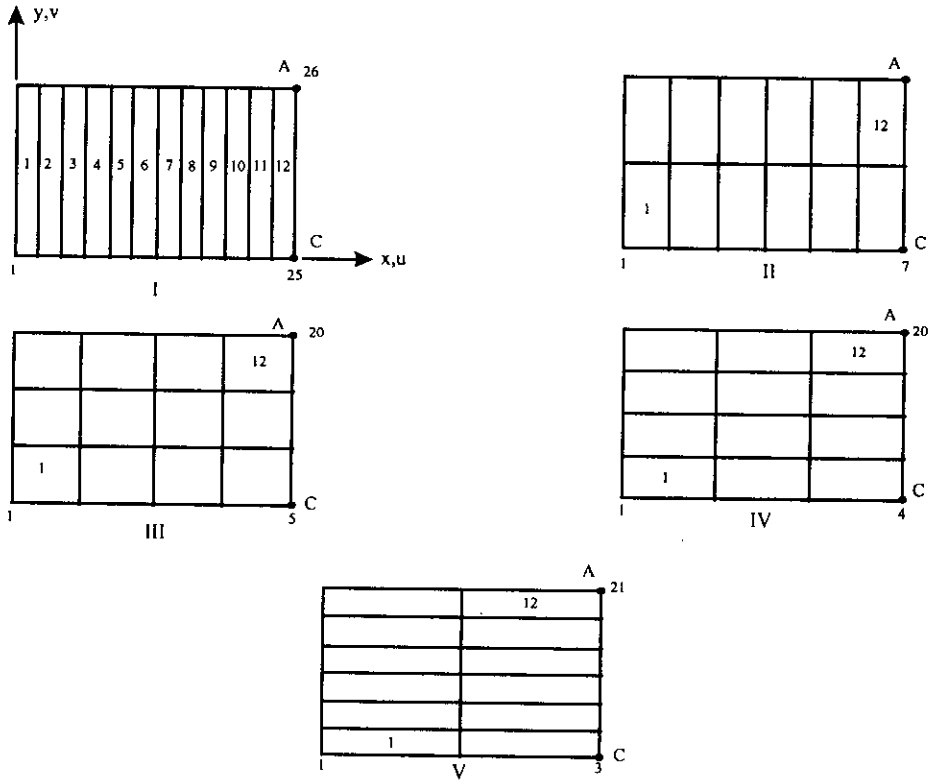



**Fig. 4.4(a): Cases with different aspect ratios**

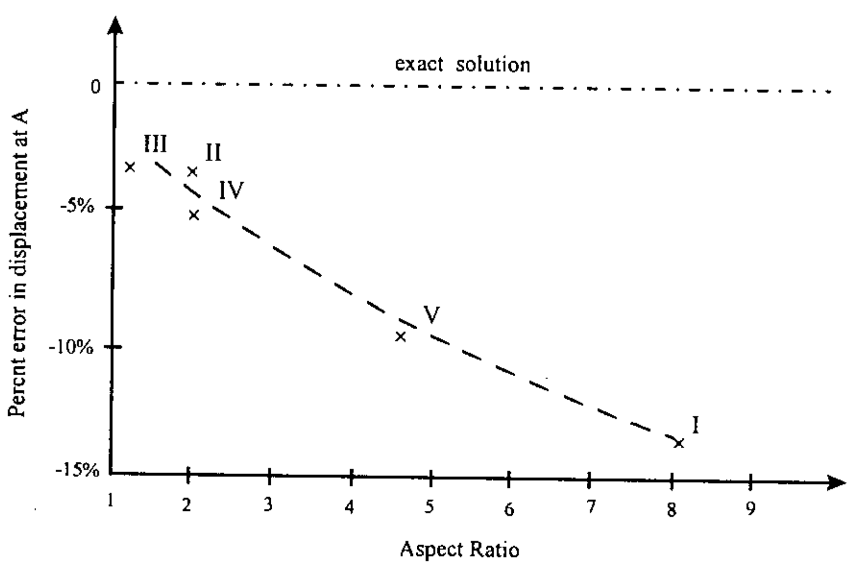



**Fig 4.4(b): Inaccuracy of solution as a function of aspect ratio**

## 4.5.4 Boundary Condition

It has been stated earlier that the idealization consists of an exterior joint with two columns and a beam framing into it. The end of the beam opposite to the end connected to the joint has been taken as free. Hence no boundary condition has been applied at this end restraining the horizontal and vertical displacements of nodes. However, the extreme nodes at this end have been coupled together to ensure that all of them underwent the same vertical displacement.

As for columns, their extreme ends which are away from the joint has been taken as fixed. Therefore, nodes at these ends have been restrained against both vertical and horizontal displacements. It can be especially mentioned here that for a two-dimensional FE analysis full fixity can be applied by resisting the translation of the structure in the direction of the two principal axes; no restraint against rotation is required.

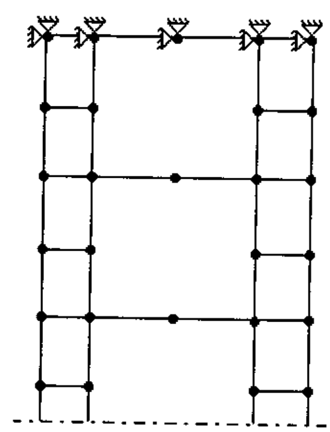


**Fig. 4.5: Boundary condition at column end**

Finally all the co-incident nodes of the rectangular concrete elements and the linear reinforcement elements has been coupled together so that at any given instance they had the same vertical displacement. This has been assured only for the beam part and was not taken care of in columns since the deformation of the columns were considered minute.

### 4.5.5 Material Behaviour

All the material properties of both concrete and reinforcement used in the analysis have been taken within the elastic range. An analysis of the idealization using the properties within the inelastic region is out of the scope of this thesis and open for future endeavor.

The following material properties were used in the modeling.

**Table 4.1: Various material properties**

| **Material** | **Modulus of Elasticity** | **Poisson's Ratio** |
|---|---|---|
| Concrete | $3000\times10^{3}$ psi | 0.17 |
| Reinforcement | $30000\times10^{3}$ psi | 0.30 |

### 4.5.6 Finite Element Model Mesh

FEM mesh is the outlines of the elements used to model the object of interest. The outline of the mesh should give the appropriate view of the object being modeled. Mesh size can vary in the analysis of a single structure. Different mesh arrangements generally give slightly varying solutions. In fact in real life problems element mesh is constantly refined to get a consistent and representative solution.

According to the literature of FEM the finer the mesh in an idealization, the smaller are the elements and the better the solution. But this too has practical constraints because a very fine mesh requires tremendous computational effort, which may justify as difficult to deliver even by the mainstream desktop computers.

Another point is that, with elements having higher order shape functions the degree of gain in accuracy diminishes after a certain level of fineness in mesh discretization. In fact there are a few elements which give exact solution even for only one-element discretization such as beam elements. Thus, it is not always necessary to use a very fine mesh at the expense of huge computer memory and computational time.

It is to be noted here that no mesh is usually the ultimate one, giving an accurate solution. A refinement of the mesh is within the scope of further studies and may be selected on the basis of its approximation of the actual result. The mesh used in the current analysis is presented in Fig. 4.6.

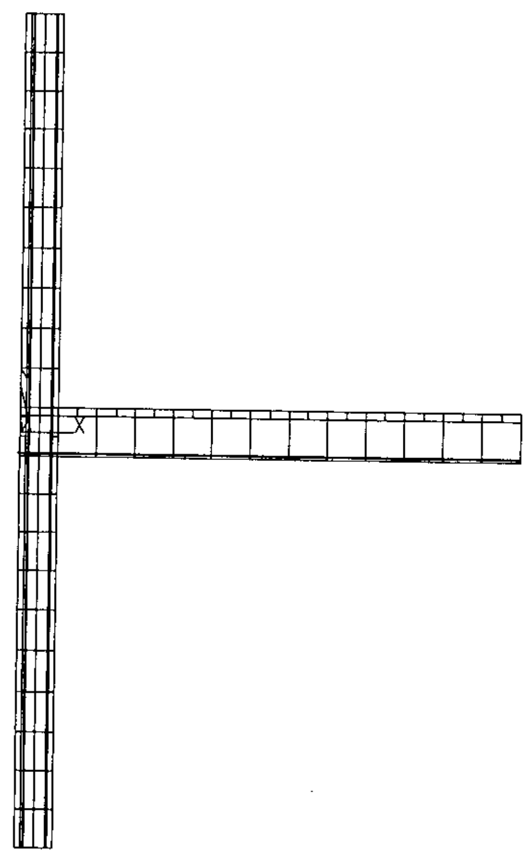


**Fig. 4.6: Finite element model mesh of the joint with beam and column**

### 4.5.7 Deformed Shapes

The following figures 4.7, 4.8 and 4.9 portray the deformed shapes of the structure. Fig. 4.7 gives the over all deformed shape of the structure. Fig. 4.8 emphasizes the deflection of the beam end and fig. 4.9 shows a detailed deformed view of the joint.

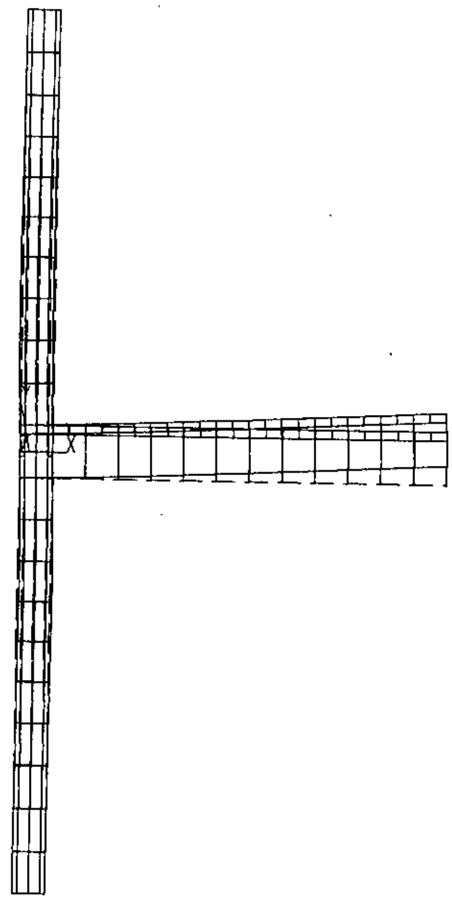


**Fig. 4.7: Deformed shape of the joint with beam and column**

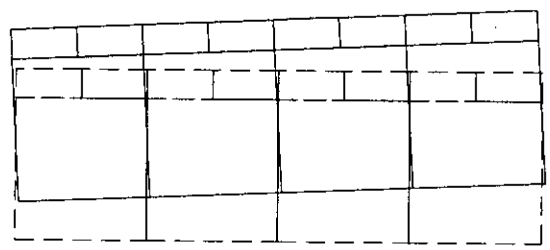


**Fig. 4.8: Deflection of the beam**

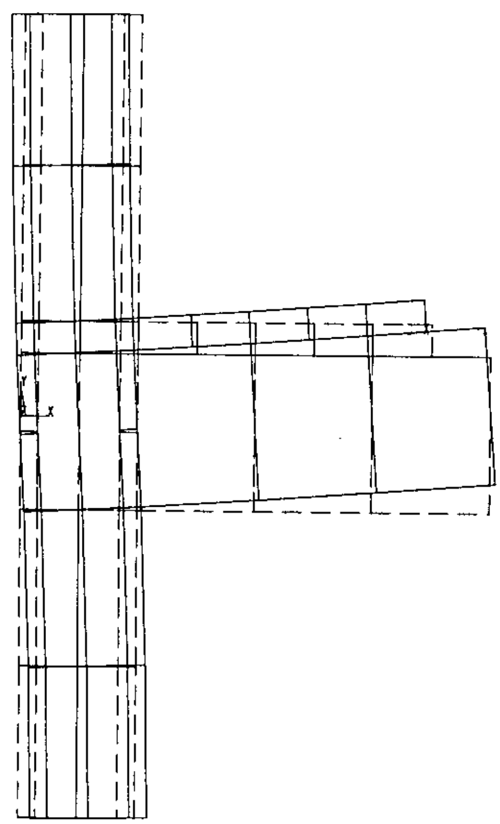


**Fig. 4.9: Deformation of the joint**

### 4.5.8 Performance of the Finite Element Model

Once the structure is idealized using finite elements, investigation should be carried out to check the performance of the model with respect to results obtained from conventional analysis. In this respect a structure is selected which can be described by the same skeletal frame, that was adopted for the analysis. Load is applied to the structure and horizontal reactions at column ends are calculated using stiffness method. This answer is then compared to the solution obtained from the FE technique. Thus this comparison will in fact provide an estimation of the degree of approximation that finite element modeling is able to achieve. However, the performance of FEM here is solely dependent on the modeling of the structure. It is subjected to variation in the hands of other people implementing the idealization in different ways.

For the comparison column size has been taken to be 12 in. × 10 in. with 2% steel reinforcement. The beam has a height of 15 in. and a width of 10 in. reinforced with 2% steel reinforcement for both tension and compression steel. The moment of inertia of the beam and column sections needed for the frame analysis are calculated according to the ACI recommended procedure. The values of Young's modulus for the materials are for concrete $3000\times10^3$ psi and for steel $30{,}000\times10^3$ psi. An upward load of $P = 5$ kips was applied at point D as shown in fig. 4.1. The horizontal reaction at top or bottom support calculated from ordinary frame analysis was 3.750 kips while the same from finite element analysis using the mesh in fig. 4.6 had a value of 3.448 kips, giving a variation of only about 8%.

## 4.6 PARAMETRIC STUDY

It was discussed earlier that the aim of this chapter is to analyze a reference structure that simulates joint rotational behaviour. So far detail discussions have been carried out on the methodologies of the approaches. Now in this section, attempts would be made to investigate the comparative analysis of the results obtained from the two approaches studied. At this point it can be reiterated that in all the cases a certain constant load at the free end of the cantilever is being applied and then the deflection of the free end is measured. This however is being done repeatedly for certain changes in the geometric properties of the beam and column sections.

### 4.6.1 Basic Approach to Parametric Study

The general idea of parametric study for a number of independent parameters embodies the fact that at a single instance only one variable should be allowed to vary while all other parameters are fixed at some initial value. If two or more parameters are allowed to vary at the same time it would cause a confusion in the results of the parametric study and their interpretation. Following this ideology, initial values for all the parameters were fixed in the study at the very outset of the parametric study.

Another point that is worth mentioning is the range of different variables. As the parameters are varied one at a time it is expected that they remain within certain bounds. This is due to the fact that exceptionally large or small values, which are not likely to occur in real-life problems, would cause wastage of computational effort. Hence investigation at hand specifies a fixed range for all the variables within which the actual work of parametric study is carried out.

### 4.6.2 Range and Initial Values of Parameters

The following table summarizes the scope and initial values of the different variables used in the parametric study. It is to be mentioned that the beam width and the column width remained constant at 10 inch throughout the whole investigation.

**Table 4.2: Initial values and range of parameters**

| Name of Parameter | Initial value | Values within range |
|---|---|---|
| Beam depth | 15 in. | 15, 18, 21, 24; |
| % of bottom reinforcement of beam | 2% | 2%, 3%, 4%, 5%, 6%; |
| % of column reinforcement | 2% | 2%, 3%, 4%, 5%, 6% |

### 4.6.3 Discussion on Parametric Study

In the following sub-sections the findings of the parametric study are discussed by referencing to appropriate graphs and plots. Eventually this parametric study will help towards the identification of differences between the conventional analysis technique and the finite element model technique. These differences, then, can be attributed to the bond-slip mechanism that imparts some rotational characteristics to the joint.

**a) Effect of variation of beam depth:** Fig. 4.10 represents the variation in deflection due to change of depth of beam. The results have been plotted for both conventional analysis and finite element analysis. The nature of the plot conforms to the general understanding. It is observed that in both analyses, the deflection of the free end decreases with an increase in the beam depth. This is due to the fact that an increase in beam depth gives rise to increased stiffness and hence reduces deflection. However an important thing that should be noticed is the rate of decrease of deflection with the increase of beam depth. In case of FE modeling it is quite higher than conventional analysis. This may be attributed to the fact that the compressibility of larger mass of concrete reduces the amount of deflection drastically. This helps the initial notion that concrete due to its highly compressive nature may in fact contribute to the semi-rigidity of joints.

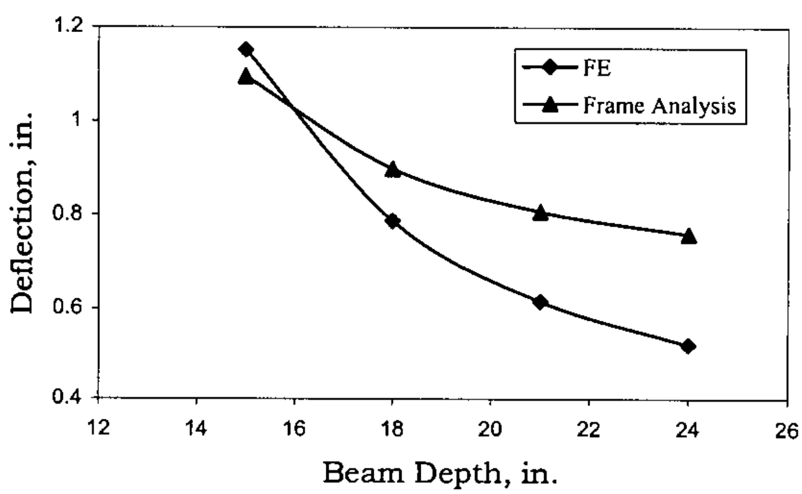



**Fig. 4.10: Deflection vs. beam depth**

**b) Effect of variation of percentage of beam reinforcement:** This effect is evident in fig. 4.11 that plots the variation of deflection with respect to change in the percentage of beam reinforcement.

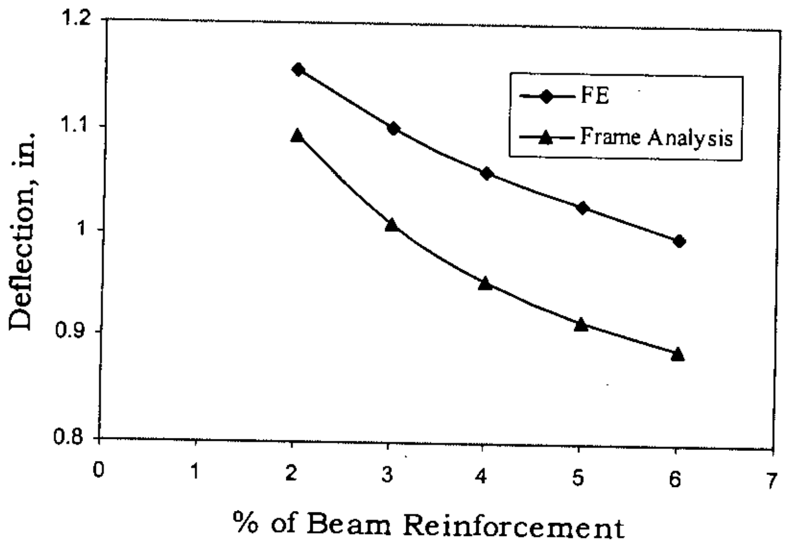



**Fig. 4.11: Deflection for different % of beam reinforcement**

Here also a decrease in deflection was observed due to an increase in percentage of beam reinforcement. This is quite logical as an increase in the beam reinforcement contributes to the increase of over all stiffness of the beam resulting in a decline in the deflection. The finite element analysis, in general, shows a higher value of deflection from the conventional analysis. This is quite all right in the light of the fact that in FE analysis bond-slip mechanism was incorporated rather than considering the reinforced concrete as a homogenous material. FE analysis thus comes closer to the real life situation and gives more reasonably accurate values. However, once again, it is proved that there is considerable amount of difference in incorporating and not incorporating bond-slip mechanism within the analysis.

**c) Effect of variation of percentage of column reinforcement:** Fig. 4.12 summarizes this effect.

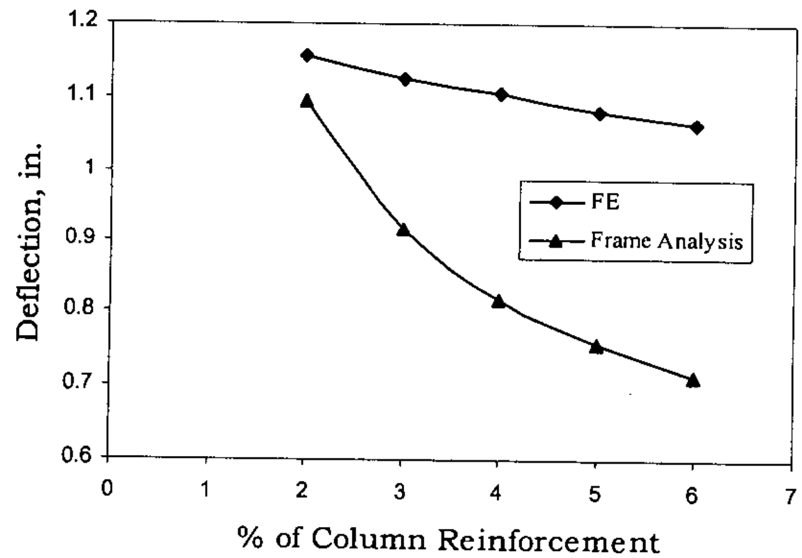



**Fig. 4.12: Deflection for different % of column reinforcement**

This effect is not that different from the variation of percentage of beam reinforcement. FE solution again shows larger deflection due to the incorporation of slipping mechanism at the interface of concrete and reinforcement. This also helps corroborate one's notion that the bond-slip in reinforced concrete may render it semi-rigid and in turn make a RC joint flexible to some degree.

## 4.7 REMARKS

The exterior joint with beam and column arrangements was analyzed using conventional frame analysis and finite element technique. From the study it was determined that the results obtained from these two analyses have differences that may be attributed to the bond-slip mechanism between concrete and reinforcement. Conventional analysis, being unable to incorporate this phenomenon gives results that differ from the finite element analysis. The presence of this property of reinforced concrete thus imparts some rotational behaviour in joints that cannot be neglected. Hence in future modeling of RC structures arrangements should be made so that this feature is addressed.

Chapter 5

# STUDY OF THE SWAY BEHAVIOUR OF RC FRAME

## 5.1 INTRODUCTION

It has been established in the previous chapter that there is significant difference in the deflections of a RC structure if it is analyzed incorporating bond-slip mechanism rather than treating the combination of cement concrete and steel as a homogenous mass. It has been suggested that there may exist some semi-rigid property within concrete. This in turn renders a RC joint somewhat semi-rigid. Conventional analysis, on the other hand, considers full slope continuity among members framing into a joint. However, this joint rotational behaviour plays an important role in the overall sway of a structure. For steel constructions, researchers have been interpreting the joint rotational characteristics with the $M$-$\phi$ relationship for sway calculation for a long time. Any investigation of the sway of the RC structure should not also neglect this phenomenon. Having established the possible presence of rotational characteristics of RC structural joints in previous chapter, attention is now turned to its effect on the lateral deflection of RC frames. Here in this chapter, efforts would be made to investigate the sway pattern of a RC frame in somewhat similar fashion as the previous chapter. Upon completion of the analysis, a comparative study would be carried out similar to the previous chapter and comments would be made on the differences if any.

## 5.2 STUDY OF A RC FRAME

As before, at the outset of the analyses, it is required to select a structure that would be subjected to the two distinct methods of analyses. For this purpose a simple portal frame would be selected at this stage. Modern-age RC building constructions, in most of the cases, consist beam and column arrangements in regular patterns. A simple portal frame is the most basic structural element of such a structure. In fact any building structure can be summarized as different arrangements of a number of portal frames (fig. 5.1). Therefore it may be justified to say that the study of the sway of a single story portal frame might be instrumental in estimating the lateral deflection of a building.

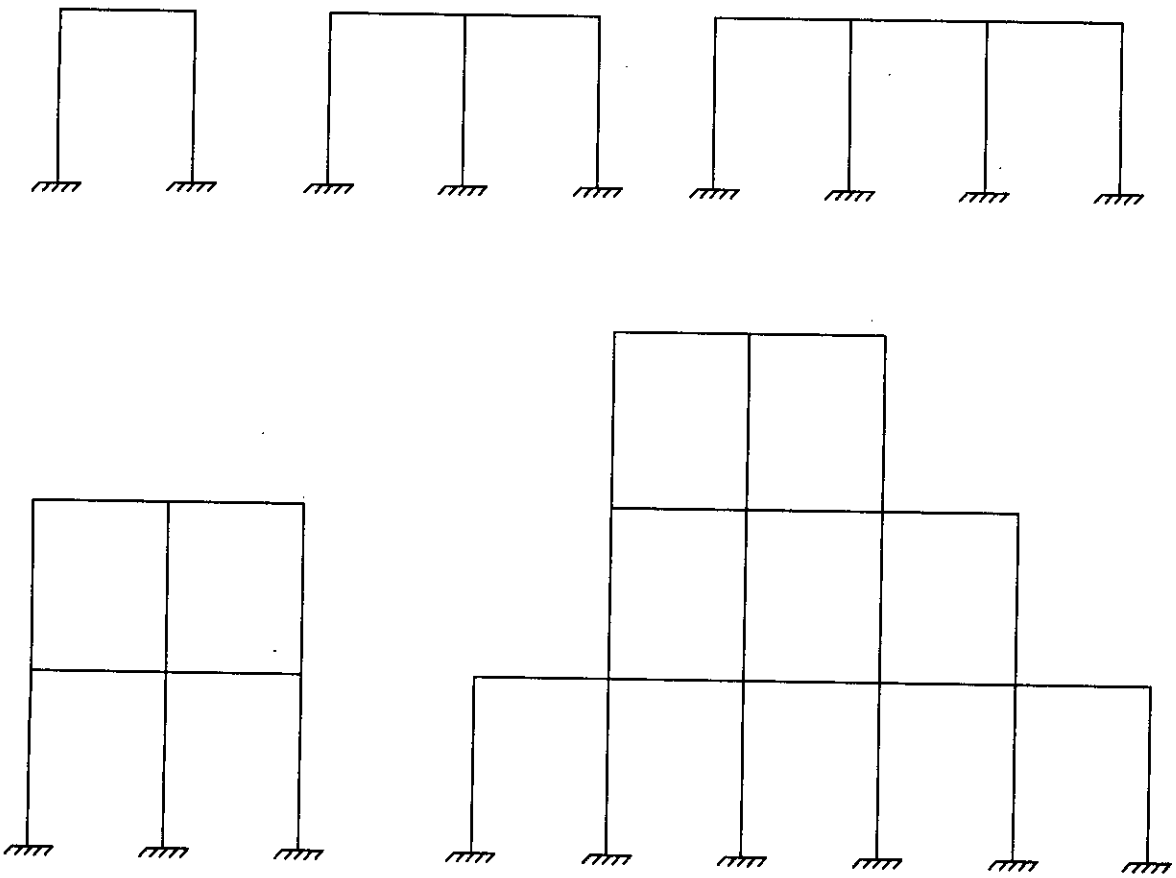


**Fig. 5.1: Different arrangement of portal frames**

The frame structure under investigation is shown in fig. 5.2. Satisfying all the requirements, it has two joints with a beam and two columns connected to them. Lower end of each column is fixed rigidly to the ground. Unlike the structure investigated in the previous chapter, most of the geometrical and material properties of this portal frame are not fixed except a few. Thus during the analyses specifications such as the beam length, column length, beam height, column thickness etc. are treated as variables.

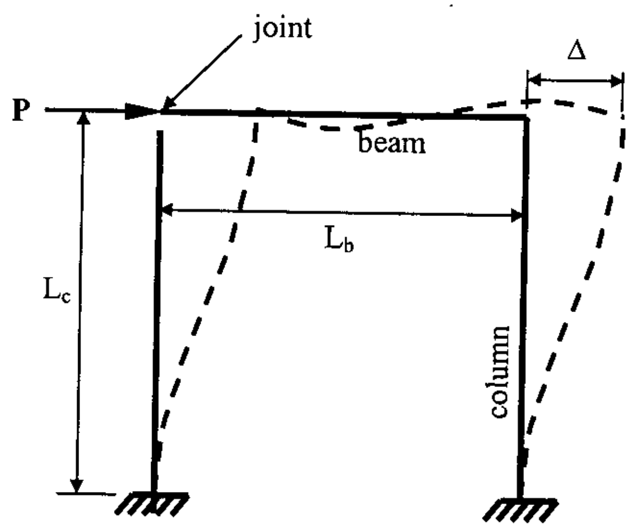



**Fig. 5.2: The portal frame**

## 5.3 METHODOLOGY

Following the methodology practiced in the previous chapter the target structure shown in fig. 5.2 would be subjected to two different methods of analysis, 1) conventional analysis using effective moment of inertia method as proposed by ACI and 2) finite element modeling using discretization technique described in chapter 3. In both cases the left joint would be subjected to a lateral load of $P$ = 5 kips. The lateral deflection of the right joint i.e. the sway of the structure would be measured. A set of sway values $\Delta$ would be obtained using variations of different geometric properties of the structures. The data thus obtained would contribute towards the comparative parametric study of the structure using two distinct methods of analysis. At the end of the study initiative should detect the effect of two analysis procedures on the sway of the structure.

Rather than discussing the variations in physical specifications of the structure at a later time, it is better to clarify it at this stage. Since the purpose at hand is to measure the differences in sway estimation using the two aforementioned analysis techniques, it would be facilitative if large number of sway data is available. Using the portal frame with a fixed set of geometric properties would only yield two values of sway for the two methods of analysis. Hence analyses were carried out on the structure for a number of times by varying the specifications of the portal structure so that considerable amount of sway values are available to establish a reasonable difference, if any. The parameters that were varied in this regard are as follow.

a) Span of the beam
b) Height of the columns
c) Depth of the beam
d) Width of the columns
e) Amount of reinforcement in beam
f) Percentage of reinforcement in columns
g) Magnitude of concrete tensile capacity

It is noted here again that simultaneous variations of all the parameters are quite impossible and would create confusion in the interpretation of results. A systemic parametric study is unlikely if all the parameters are changed at the same time. The methodology adopted here is that, at any time all parameters are fixed except one. This one parameter is then assigned different values and corresponding analyses are carried out to calculate the sway. Once the investigation of this parameter is finished, it assumes a fixed value like the others and a new parameter now becomes variable. This process is thus continued throughout the entire study.

Following the discussion above, it can be deduced that the various parameters must have a constant value which each of them assume while some other parameter is varied. The following table describes the fixed value of the parameters along with their range of variations.

**Table 5.1: Variation in geometric parameters of the portal frame**

| Name of parameter | Constant value | Values within range |
|---|---|---|
| Span of the beam | 16 ft. | 12,16, 20, 24, 28 |
| Height of columns | 10 ft. | 9, 10, 11, 12 |
| Depth of the beam | 20 in. | 12, 16, 20, 24, 28 |

| | | |
|---|---|---|
| Depth of the columns | 16 in. | 12, 16, 20, 24, 28 |
| Amount of reinforcement in beams | $\rho_{avg}$ | $\rho_{\min}, \dfrac{\rho_{\min}+\rho_{avg}}{2}, \rho_{avg}, \dfrac{\rho_{avg}+\rho_{\max}}{2}, \rho_{ma}$ |
| % of reinforcement in columns | 3% | 1%, 2%, 3%, 4%, 5% |
| Concrete tensile capacity | $4\sqrt{f_c^{'}}$ | $3\sqrt{f_c^{'}}, 4\sqrt{f_c^{'}}, 5\sqrt{f_c^{'}}$ |

The points discussed below should also be noted.

- The width of the beam and the depth of the column are not within the variable parameters and their values have also not been stated. Since the finite element modeling involves two-dimensional plane stress element the thickness of these elements has been considered fixed. And since the width of the beam and the depth of the columns are represented by this plane stress element thickness, there was no scope of varying this amount. Thus this amount assumes a constant value of 12 inch for all of the analyses i.e. the depths of the columns and the beam widths are always kept at 12 inch.
- Regarding the amount of reinforcement in beam, same value has been assigned for both top and bottom reinforcement of the beam. In other words, for a certain analysis, if the amount of bottom reinforcement was $\rho_{avg}$ then the top reinforcement also was fixed at $\rho_{avg}$.
- The columns had an arrangement of reinforcements as shown in fig. 5.3. The percentage of reinforcement calculated was hence distributed equally among the three layers of main reinforcements in columns.

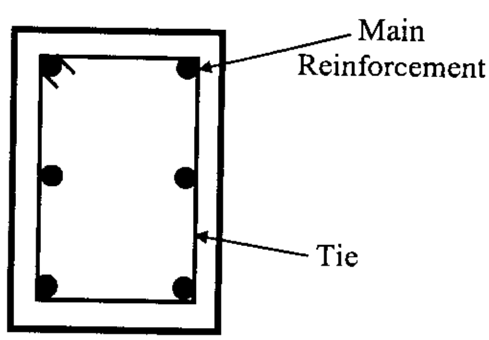



**Fig. 5.3: Arrangement of reinforcements in columns**

## 5.4 ANALYSIS USING ACI GUIDELINE

The ACI effective moment of inertia method of analysis is treated as the conventional analysis technique in this thesis. As described previously the analysis is carried out following the guidelines presented in ACI committee report 435R-95 and using the refinement as described in chapter 2. The portal frame is subjected to a lateral load of 5 kips and the sway of the frame is calculated. For consecutive analyses one physical parameter is varied while others are kept constant at certain median value. It is to be noted, however, that following the discussion of chapter 2, for a single calculation of the sway the portal frame has to be analyzed for a number of times, so that all the values of actual moment $M_a$ and the effective moment of inertia $I_e$ for all columns and the beam converge in consecutive runs. Therefore it is required that the frame is analyzed for a number of times before decision is taken on the final sway value for a particular parameter. In fact in the current study, the frame was analyzed for several times to calculate a single value of lateral deflection. This onerous repetitive task was performed using spreadsheet software where a detailed worksheet was made, that analyzes the portal structure several times whenever a certain value of any parameter is changed. Using the worksheet the rate of tolerance between consecutive values of $M_a$ and $I_e$ was found to be 0-2%, which is quite good. It is worthy of mentioning that the sway estimation was done using stiffness method programmed within the spreadsheet software.

## 5.5 FINITE ELEMENT ANALYSIS

Finite element modeling of the portal frame was carried out using the discretization guideline expressed in chapter 2. The concrete was modeled using two-dimensional 4 noded and 8 noded quadrilateral elements while reinforcements were represented using one-dimensional link element. The phenomenon of bond-slip was incorporated within the modeling by putting special dimensionless spring elements at the connection of concrete and reinforcements. The structure was then subjected to the same lateral load of 5 kips and was analyzed to estimate the sway of the structure. Like conventional technique, here too, all the parameters except one were kept constant and thus the parametric study was carried out. However, some points special to finite element modeling and analysis should be emphasized which are discussed in the following sub-sections.

### 5.5.1 Advantage over Conventional Analysis

Since during the finite element modeling 2-dimensional plane stress elements are being used, shear deformations as well as flexural deformations are automatically incorporated in the model analysis. The deformation results, therefore, shall automatically reflect the contribution of shear and bending deformation, which is otherwise not possible to simulate in conventional frame analysis. Thus the sway calculation from the finite element modeling are likely to be more realistic.

### 5.5.2 Modeling of Beam and Column

Unlike the beam modeled in the previous chapter, the beam of the portal frame was modeled to its full entirety. The structure modeled in chapter 4 assumed that, due to cracking the concrete at the tension face is rendered unnecessary and hence did not provide any element to simulate concrete below the level of effective depth. This however was not the case in analyzing the portal frame. Here the full of the beam depth was simulated using 4 noded and 8 noded quad elements. Also, since each column would be subjected to tension and compression at the same time, they were also modeled to their full depth using elements. The figures 5.4 and 5.5 show details of the modeling of the beam and column respectively.

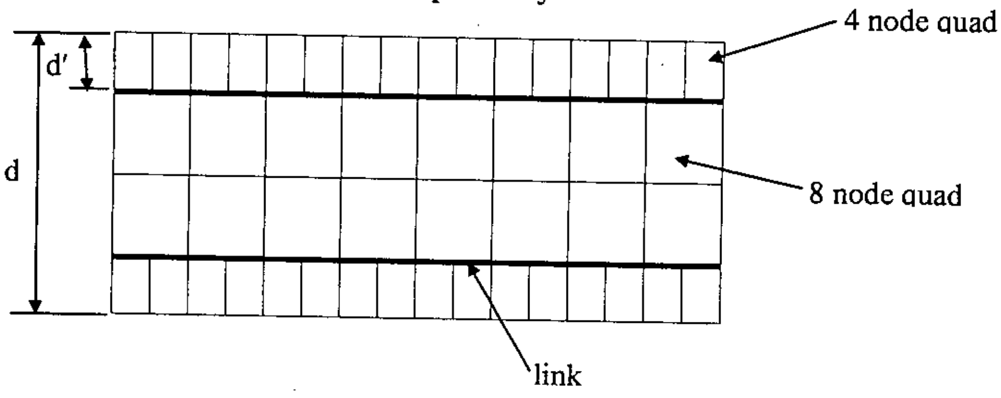



**Fig. 5.4: Details of beam modeling**

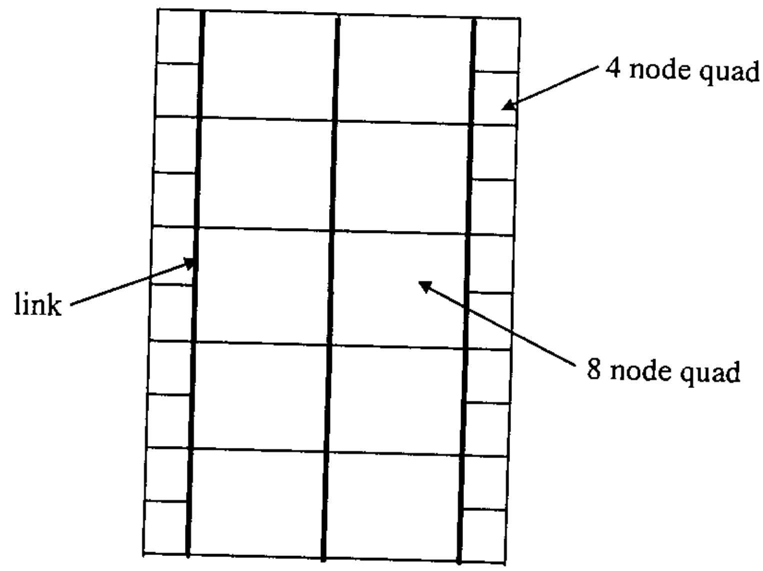



**Fig. 5.5: Details of column modeling**

## 5.5.3 Boundary Conditions

Boundary conditions that were applied to the finite element model were two folds. As can be seen from fig. 5.2 that the ends of the columns are rigidly connected to the ground. Therefore one set of boundary conditions were applied to restrict the degrees of freedom at the nodes of the columns adjacent to the ground. This is clearly evident from the fig. 5.6 representing the subsection of the model mesh near the column boundaries.

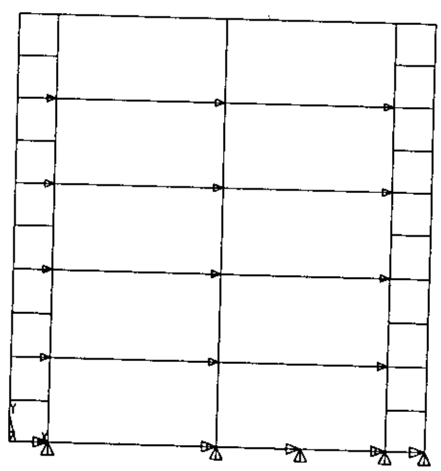


**Fig. 5.6: Boundary condition at column ends**

The other set of boundary conditions was applied at co-incident nodes in beams and columns. It can be remembered that the reinforcements are connected to concrete using special dimensionless spring elements simulating the bond-slip mechanism. This requires that two coincident nodes be placed wherever a spring is needed. However these springs have two degrees of freedom, meaning that they can have horizontal and vertical translations at the same time. But this is not desirable in the present situation. Because bond-slip in beam would only allow horizontal movement of the spring i.e. it would allow the reinforcement to move laterally within the beam. The same is true for columns where the reinforcements should be allowed to move only in the vertical direction following the idea of bond-slip mechanism. Therefore some mechanism is needed to restrict the translations of the springs in lateral and vertical directions for beams and columns respectively. This can only be achieved if boundary conditions are applied at the coincident nodes containing the springs. Hence in the finite element model, the degrees of freedom in the vertical direction for nodes in the beam have been coupled. This would ensure that the two coincident nodes would have the same translation in the vertical direction restricting any relative movement of the spring in the same direction. Likewise, for the coincident nodes of the columns, they were coupled in the horizontal direction so that the spring connected to that may not suffer any translation in that direction. The coupling of the coincident nodes are shown in fig. 5.7 that represents a subsection of the meshing applied to the beam.

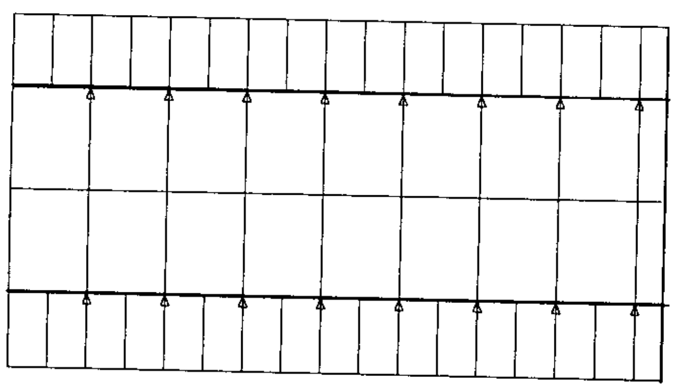


**Fig. 5.7: Coupling of coincident nodes**

### 5.5.4 Incorporation of Cracks

Unlike the discretization technique adopted in the previous chapter, the finite element model of the RC portal frame includes a special feature, which is the incorporation of cracks within the model. It is known that concrete is very weak in tension having a tensile stress roughly equal to one tenth of its capacity in compression. As lateral load is applied on the structure some of the sections of the beams and columns would be subjected to large tensile stresses. These stresses would cause considerable amount of cracks in more critically stressed sections. Fig. 5.8 illustrates the phenomenon in more detail and shows the sections of the various members of the structure that are likely to suffer from this cracking.

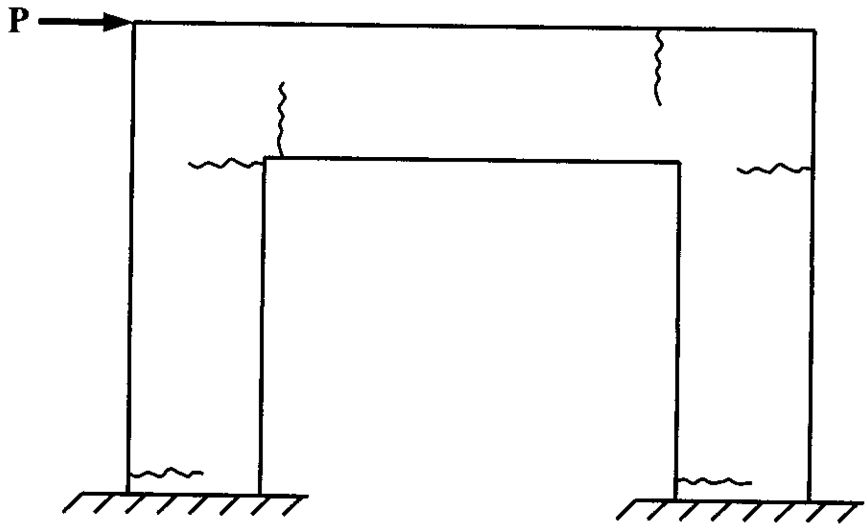



**Fig. 5.8: Possible places of cracks in portal frame**

Since the aim of modeling is to simulate the real life situation as closely as possible arrangements were made to incorporate these cracks within the modeling. To do so, sections were first identified that are prone to cracking. Generally these are the sections at beam-joint or column-joint interfaces. Then the faces of the sections were identified where cracks would occur. The general understanding of the deflection pattern of the portal frame was employed for this job. Once it was decided on which face cracks should occur, special arrangements were made to take care of it. To simulate the cracks, coincident nodes were introduced at those sections. Some of the nodes were connected to the joint elements and the others were connected with beam or column elements. The coincident nodes were then coupled both in horizontal and vertical directions. This was done for the face at which there would be no cracks. It ensured that at these faces, the coincident nodes would be connected to each other and would have same displacements, which in other term implies that there would be no cracks at these locations. On the other hand coincident nodes were not coupled at faces where cracks would occur. Since these nodes were not coupled, it means that each of them may have displacements in any direction independent of the other. Thus after the analysis these nodes may suffer different amount of displacements opening up the section at those locations and hence simulating cracked behaviour.

### 5.5.5 Finite Element Model Mesh

Actually all the specific details of the finite element model mesh have been covered already in the previous sections. Here in this section a full mesh of the portal structure is presented in the following figure. Also fig. 5.9 represents the mesh of a beam and column joint of the RC portal frame.

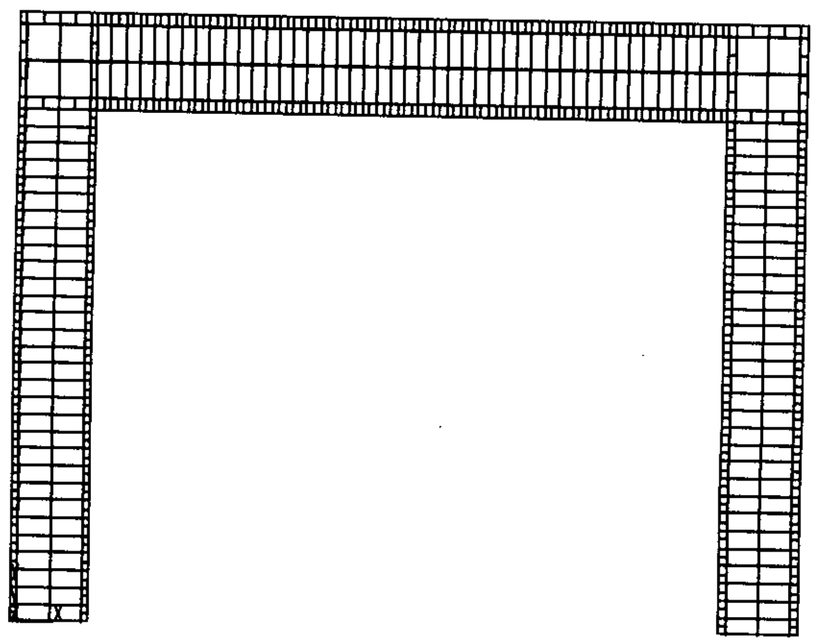


**Fig. 5.9: Mesh of the portal structure**

## 5.5.6 Deformed Shapes

In this section, the deformed shape of the portal structure is presented in fig. 5.10. It is observed that the deformed shape of the structure clearly resembles the one shown in fig. 5.2. Also Fig. 5.11 shows the cracks produced at the left joint. It is also to be noted that these cracks developed within the joint where they are expected to occur. Thus assumption about the location of cracks is confirmed. Finally fig 5.12 shows the crack at the interface of the left joint and the beam in more detail.

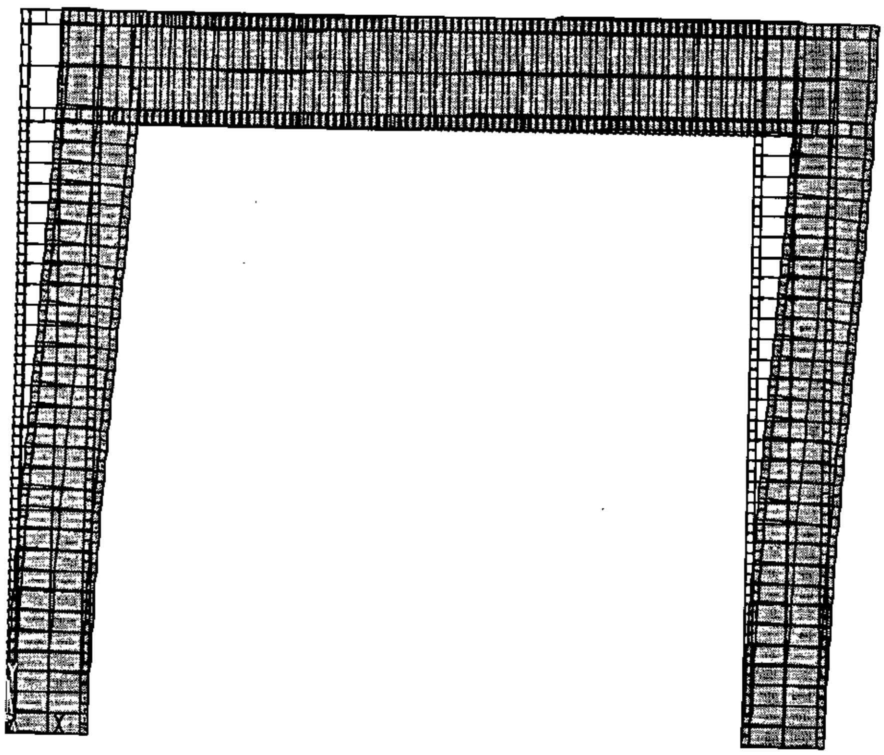


**Fig. 5.10: Deformed shape of the portal frame**

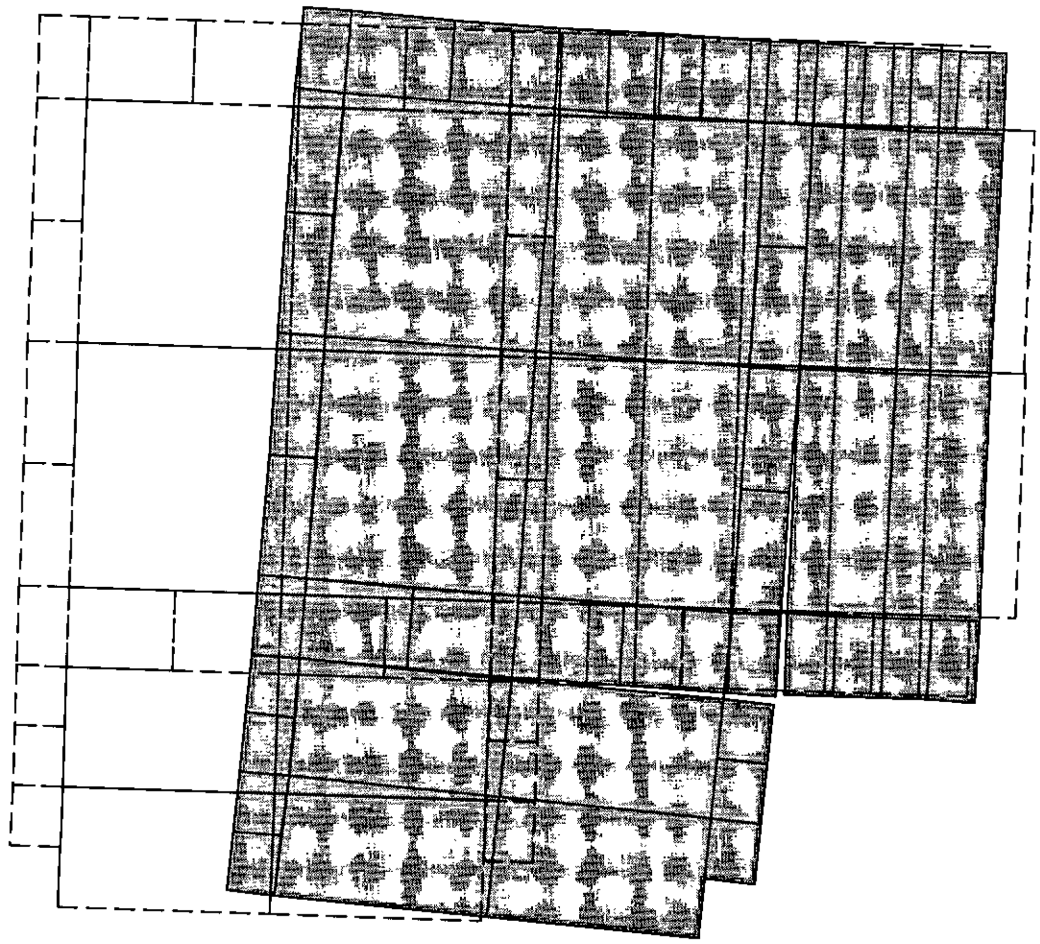


**Fig. 5.11: Cracks developed at left joint**

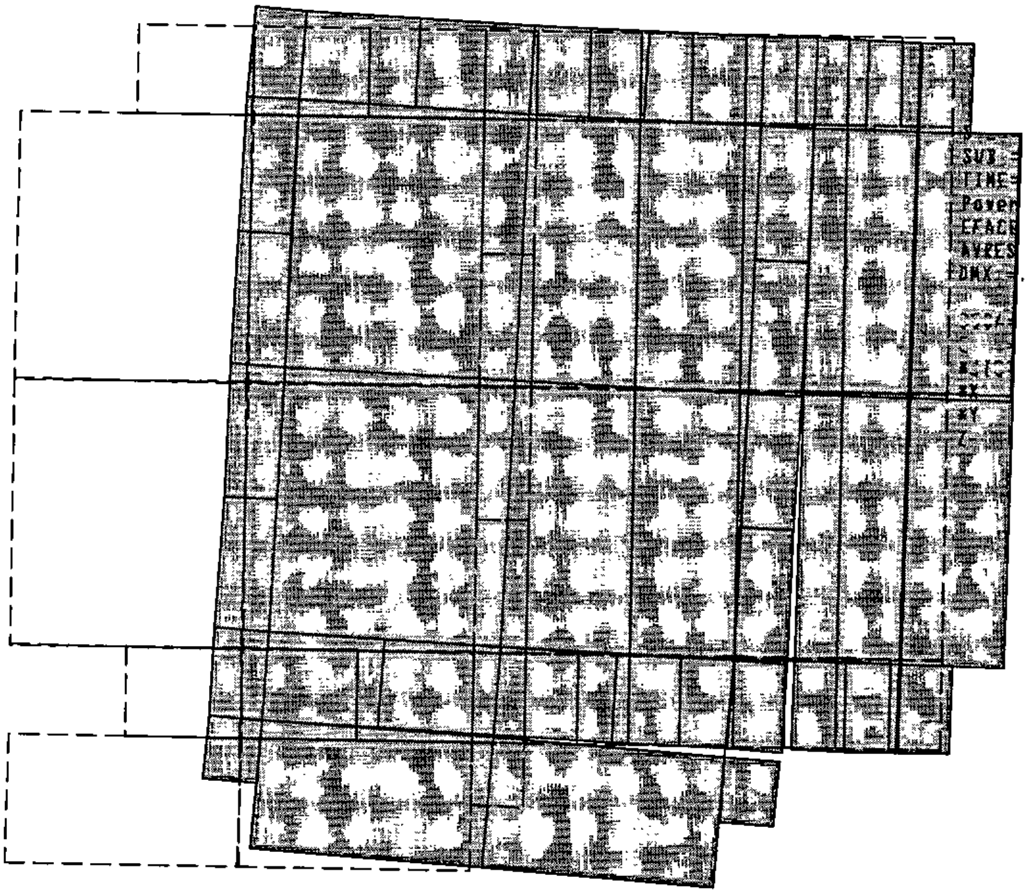


**Fig. 5.12: Details of cracks at left joint**

## 5.6 PERFORMANCE OF THE FINITE ELEMENT MODEL

Having discussed the essential details of the finite element modeling, checks should be performed on the model to determine its representativeness. One way to do this is to simulate a real life portal frame with the model. Then the real life portal structure and the FE model should be subjected to same amount of lateral loads and corresponding deflections should be measured. If the sway value for the FE model comes closer to the actual test model then the FE model can be considered as sufficiently representative. Since it was out of the scope of the current study to test a real life model of a portal frame, it was impossible to check the performance of the FE model by direct testing. However, an indirect performance checking was carried out by comparing the results obtained from FE model with those obtained from the study of Mehrabi, Shing, Schuller and Noland (1994). As discussed in chapter 2, their investigation was based on study of an actual model. Comparing these two sway values the graph in fig. 5.13 was obtained. It is observed that the FE model yields a sway value reasonably close to the actual test model. Also evident from the graph is the fact that sway calculated for the same structure using ACI method yields a value much larger than that of the actual test. Therefore, it can be concluded that the FE model of the current study has good performance with respect to actual test model of a simple portal frame.

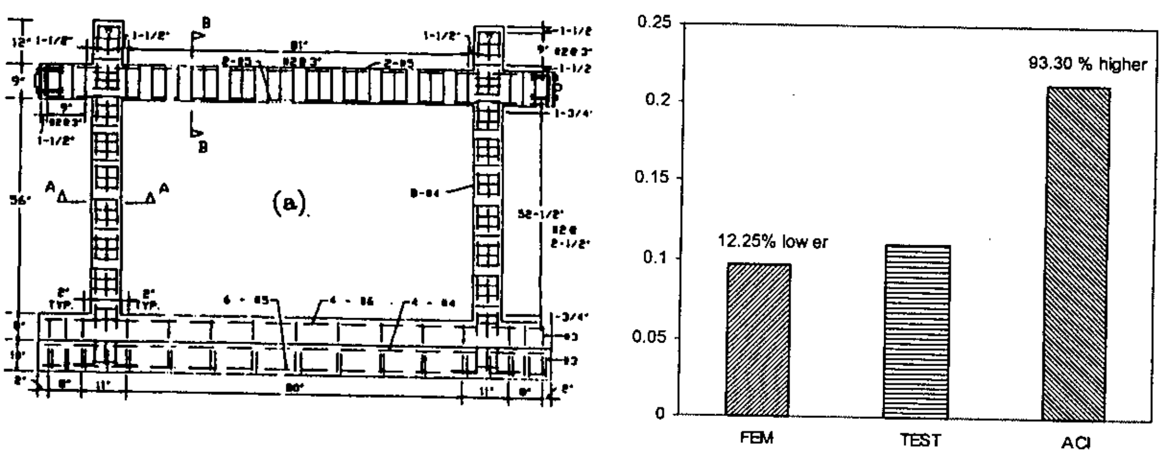



**Fig. 5.13: Test model and sway results comparison**

## 5.7 COMAPRATIVE ANALYSIS

Once the analyses are performed attempts are made to perform parametric study of the structure. But the approach of this analysis would be somewhat different than the one carried out in the previous chapter. Here the aim is not to identify the effect of various parameters on the sway of the RC structure but to find out the difference in the sway values obtained from the conventional ACI effective moment of inertia method and the detailed finite element modeling. If such a difference is established it can be concluded that a refinement of the ACI technique is in the order since the finite element modeling has been devised to simulate the actual situation to the highest level. The following figures show the comparative plots of the sway obtained for variations of different geometric properties of the structure..

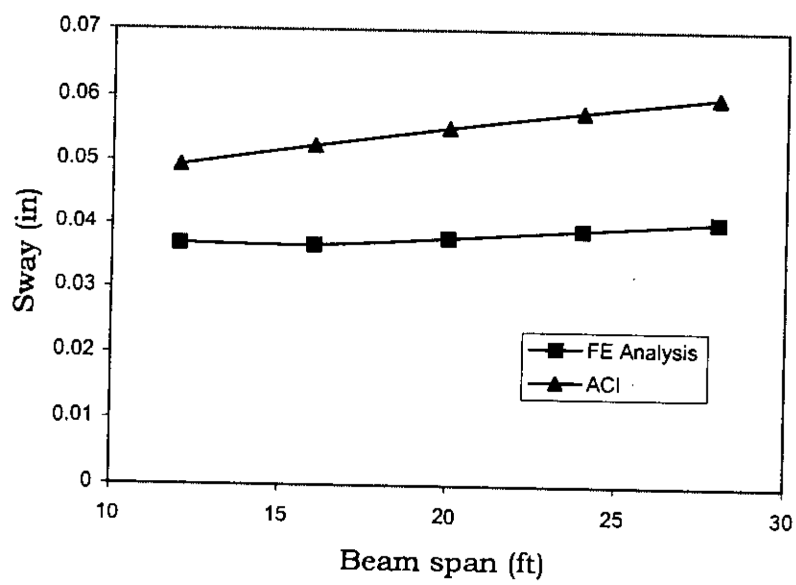



**Fig. 5.14: Sway vs. beam span**

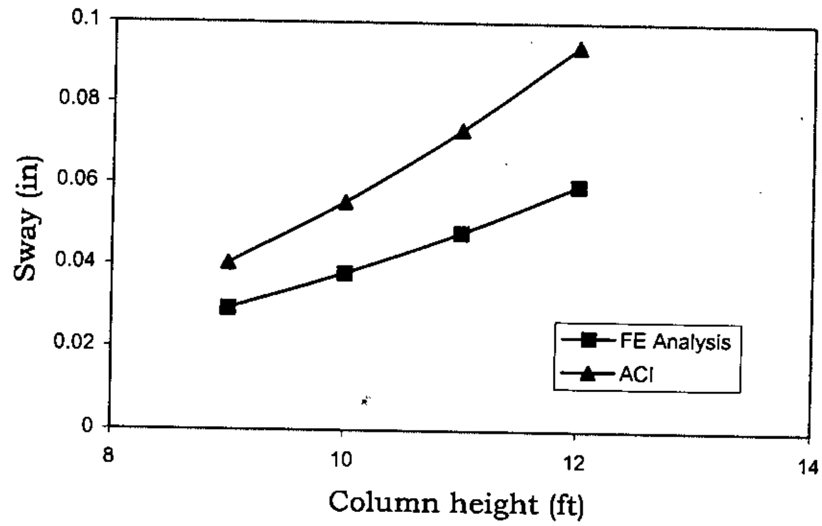



**Fig. 5.15: Sway vs. column height**

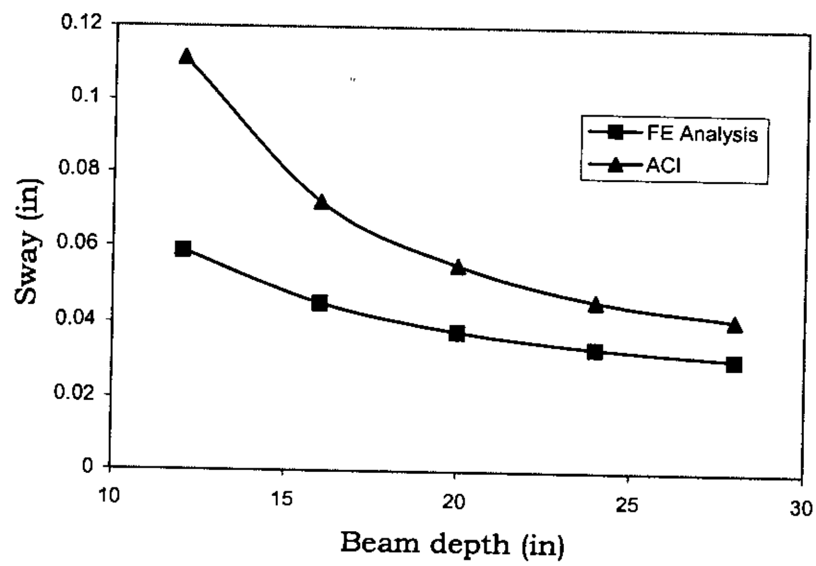



**Fig. 5.16: Sway vs. beam height**

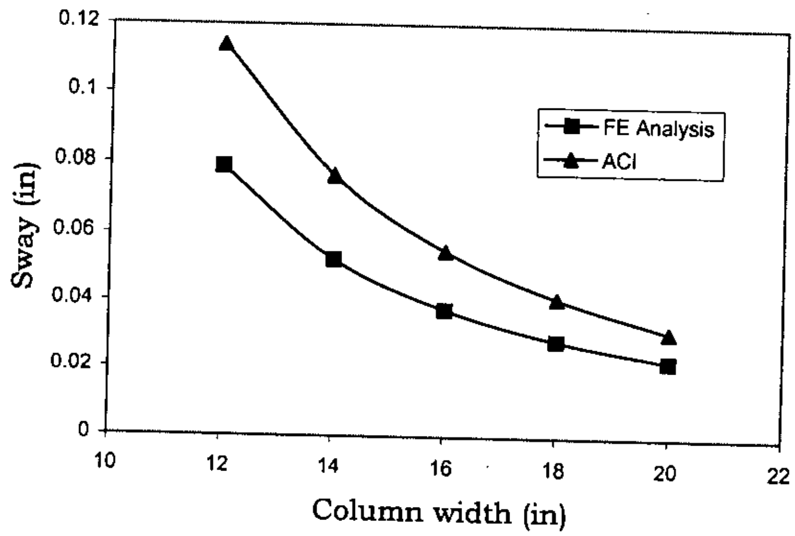



**Fig. 5.17: Sway vs. column width**

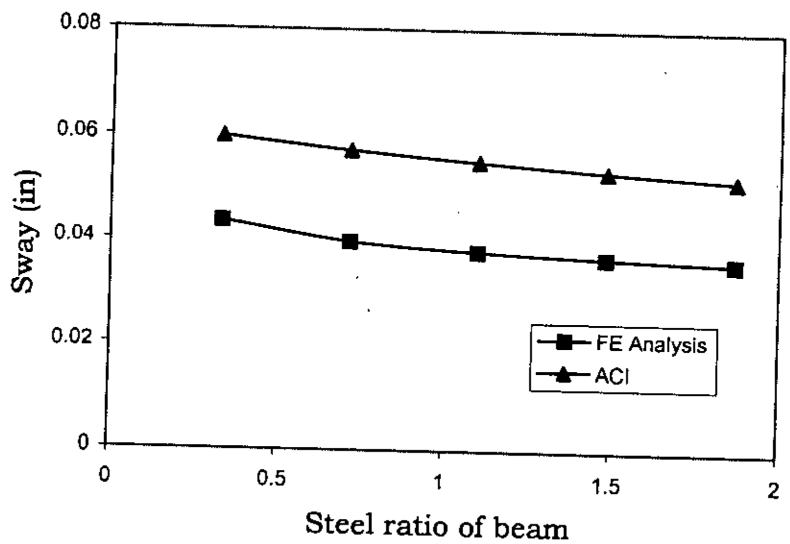



**Fig. 5.18: Sway vs. ρ of beam**

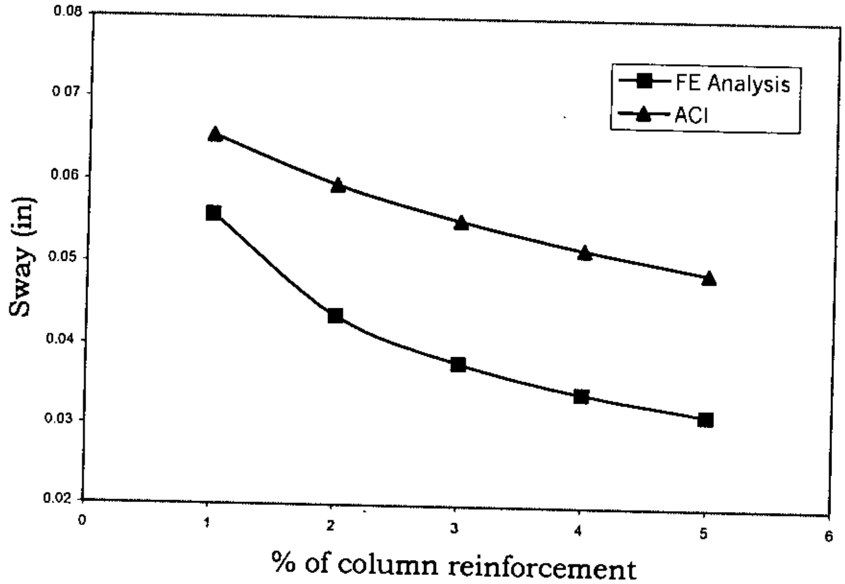



**Fig 5.19: Sway vs. % of column reinforcement**

Considering the above figures one can see that there is significant difference in the sway value calculated from finite element analysis and effective moment of inertia method of analysis. This difference in extreme cases ranges beyond 200% more than the lower value. It is also observed that in all the cases the values calculated using finite element technique are lower than those calculated by ACI proposed method. This implies that if the effective moment of inertia method is adopted in regular design procedures, sway values would be over estimated, which in turn means that larger sections must be provided resulting in less economy.

## 5.8 REMARKS

This chapter succeeds in establishing the fact that the sway of the RC structure is influenced by the joint rotational behaviour characteristics imparted by the bond-slip mechanism at the interface of concrete and reinforcement. Traditional analysis does not take this fact into consideration and hence over estimates the sway of structures. This situation should be avoided towards better economy. Efforts are made therefore in next chapter to refine the conventional analysis so that the sway values calculated comes closer to those obtained from finite element analysis.

## Chapter 6

# A RATIONALE FOR SWAY ANALYSIS

## 6.1 INTRODUCTION

Analyses performed in chapter 5 reveals the fact that there is significant difference in deflection pattern of a RC structure while subjected to conventional analysis and finite element modeling. The variation of the conventional analysis results from those obtained from the finite element discretization was attributed to the presence of joint rotation behaviour, which is neglected in the former method of analysis. This joint rotation characteristic is primarily related to bond-slip phenomenon present at the interface of the concrete and reinforcement. Conventional analysis, which in this case is frame analysis using ACI effective moment of inertia method, is incapable of incorporating this mechanism. Hence deflection values were obtained that are different from FE Modeling. Chapter 5 also substantiates this fact. In this chapter it was observed that the sway of a RC portal frame varies largely from the finite element analysis when analyzed using the conventional method. Joint rotation behaviour being an important factor in the sway of a structure, neglecting this phenomenon from the analysis influences the sway value to a great extent.



There is, however, no scope of abandoning the conventional analysis altogether and adopt the finite element method for all regular analyses. As described in chapter 1, finite element analysis requires certain levels of expertise and special knowledge of the subject matter. Also it could be a complicated process, which is not suitable for analysis of a large number of structures in design offices. The intricacy develops exponentially with an increase in the size of the structure. Hence some simpler methods are required that would be easier to grasp, yet have an acceptable accuracy compared to the finite element solution. With this goal in mind, the conventional analysis technique proposed by ACI might be modified to attain a level of accuracy that can reasonably compare with the finite element technique. At its current proposition the ACI method yields results too different from the FE modeling. But there are scopes of modification within the method. The purpose of this chapter, therefore, is to identify these scopes of modifications and propose the required changes. Efforts are then made to test the performance of the suggested proposition by applying the modifications to RC structural models other than the simple portal frame analyzed in chapter 5.

## 6.2 SCOPES OF MODIFICATION

To identify the areas of modifications of the ACI effective moment of inertia method the details of the technique must be reiterated a little. As described in chapter 2 the central idea of the method revolves around the calculation of effective moment of inertia of the beam and column sections using the following equation 2.4.

$$I_e = \left(\frac{M_{cr}}{M_a}\right)^3 I_g + \left[1 - \left(\frac{M_{cr}}{M_a}\right)^3\right] I_{cr}$$

This equation is then backed by the $I_e$ weightage combination formula (equation 2.5) that combines the effective moment of inertia of all the sections of a beam and column to obtain the ultimate $I_e$ of that member.

$$I_e = 0.25(I_{e(1)} + I_{e(2)}) + 0.5 I_{e(m)}$$

The equation 2.4 can be written using a more generalized form as shown below.

$$I_e = \left(\frac{M_{cr}}{M_a}\right)^\alpha I_g + \left[1 - \left(\frac{M_{cr}}{M_a}\right)^\alpha\right] I_{cr} \qquad 6.1$$

where the value of $\alpha$ is 3.

Likewise the generalized form of equation 2.5 would be as follows.

$$I_e = \omega(I_{e(1)} + I_{e(2)}) + (1 - 2\omega)I_{e(m)} \tag{6.2}$$

Here $\omega$ has a value of 0.25. ACI however is somewhat flexible about the value of $\omega$ and suggests other values like 0.15 for different end conditions.

Basically, these two equations 6.1 and 6.2 embody the main idea of the method proposed by ACI. Now a close observation of these two equations reveals that there are some possible areas that can be modified to obtain sway values with greater accuracy. Also there is another point to observe. If the guidelines of ACI Committee Report 435R-95 are followed, then the report recommends gross moment of inertia, $I_g$ to be calculated as $bh^3/12$, where $b$ is the width of the section and $h$ is the height. This estimation of $I_g$ completely ignores the contribution of the steel reinforcement towards the gross moment of inertia of total uncracked section. The current investigation, therefore, identifies the following three points which may be subjected to modification in attaining this goals.

a) The weighing coefficient $\omega$ used in equation 6.2 to combine $I_e$ of different sections and obtain ultimate $I_e$ of the member.
b) The numerical power $\alpha$ of the factor $M_{cr}/M_a$ used in the equation for $I_e$ calculation i.e. equation 6.1.
c) The calculations of gross moment of inertia of the section, $I_g$.

Following this assumption investigation would be carried out whether these three modifications help in raising the accuracy level of the sway calculation using this method. The methodology adopted here would be to select any of the item (a) or (b), modify its value and then to compare the results obtained with that of the finite element analysis. As for item (c) attempts would be made to suggest a different approach for calculation of $I_g$ and include it in the investigation of item (b). In fact item (b) and item (c) would go hand in hand in the current investigation and their study has been grouped together under section 6.4. The complete investigation is a trial and error process. Once the final value of the items of modification are decided, depending on the proximity of sway values with the FE solutions, attempts would be made to suggest a modified form of equations 6.1 and 6.2 incorporating the required changes.

## 6.3 STUDY OF WEIGHTAGE COMBINATION FACTORS

Before starting with investigation, equation 6.2 must be analyzed first. Equation 6.1 calculates the effective moment of inertia, $I_e$ of a section provided the actual moment $M_a$, cracked moment $M_{cr}$ and the cracked of moment of inertia $I_{cr}$ of the section is known. Now a beam or a column may have a lot of sections with varying values of $M_a$, $M_{cr}$ and $I_{cr}$ that would result in $I_e$ value of a section different to other sections of the members. Therefore a situation arises where there are lots of different $I_e$ values for a single structural member. Now the question may occur, which $I_e$ value to use during the frame analysis. To help with this dilemma ACI has proposed the equation 6.2. A close observation of this equation reveals that its purpose is to combine the $I_e$ values at the midspan and two ends of a member to obtain the over all $I_e$ of the member. For this purpose different weighing factors $\omega$ are used to combine the $I_e$ values. In equation 6.2, it is observed that the effective moment of inertia of the midspan is given the highest weightage while the ones of the two ends are given less weightage. Now there is a scope of modification of these weighing factors and it has been identified in the previous section. Even the ACI committee report 435R-95 also suggests variations of these weighing coefficients in various cases. According to this report equation 6.2 should be modified to the following equation when applied to beams continuous on both ends.

$$I_e = 0.15\left(I_{e(1)} + I_{e(2)}\right) + 0.70 I_{e(m)} \qquad 6.3$$

Also it suggests that for beams continuous on one end only the modified equation 6.4 should be used.

$$I_e = 0.85 I_{e(m)} + 0.15\left(I_{e(1)}\right) \qquad 6.4$$

So it can be deduced that there is scope of studying the weighing factor $\omega$ of equation 6.2.With this in mind efforts would be made to modify the values of the weighing coefficients of the equation. Different values for these factors have been applied and the resulting sway of the simple portal structure has been calculated. The sway values then were plotted against those obtained from the finite element analysis. The plot resulted in the following graphs.

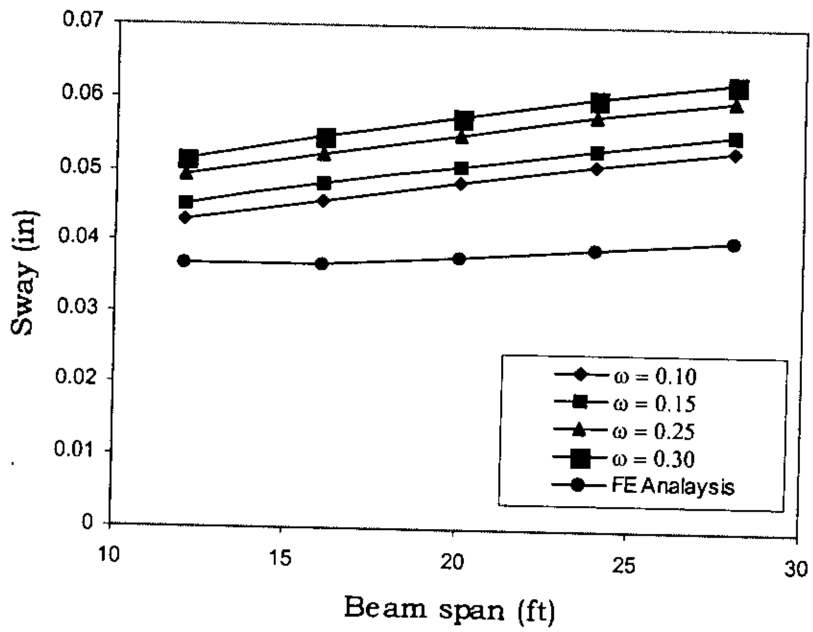



**Fig. 6.1: Sway vs. beam spans (ft) for values of ω**

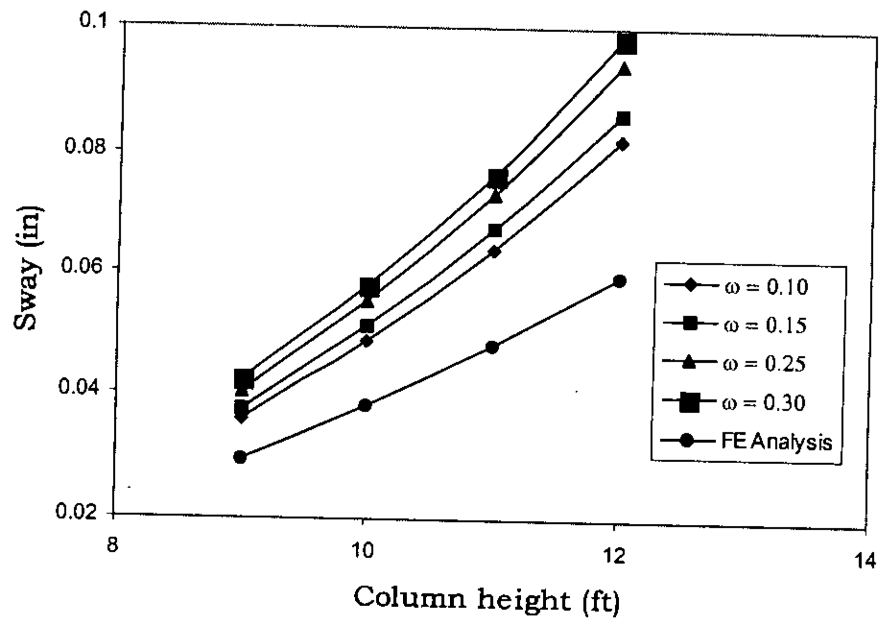



**Fig. 6.2: Sway vs. column heights (ft) for values of ω**

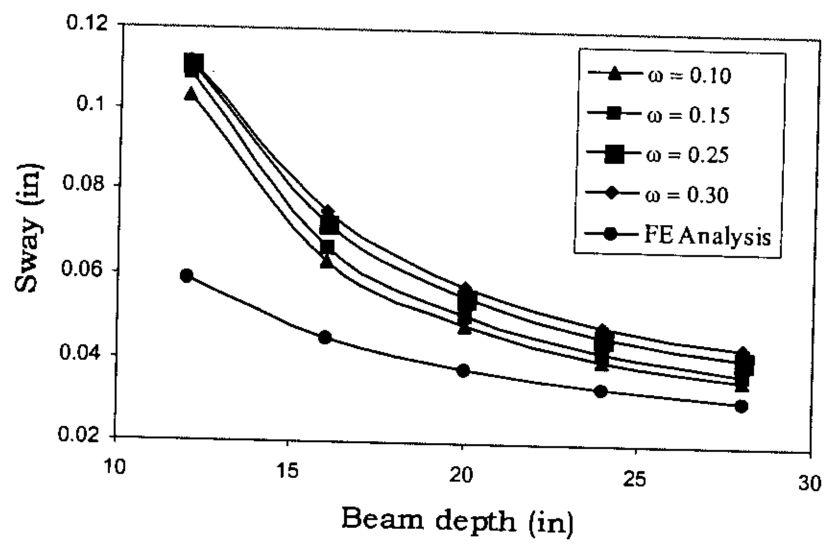



**Fig. 6.3: Sway vs. beam depths (in) for values of ω**

Fig. 6.1, 6.2 and 6.3 represents the sway pattern obtained for the variation of weighing coefficient $\omega$ for the end spans. Taking the different values of $\omega$ such as 0.10, 0.15, 0.25 and 0.30 sway calculations were performed. Analyzing the plots some common phenomenon is observed. Firstly one can see that modifying the values of $\omega$ gives plots of sway values which are fairly parallel to each other. Secondly, there is no significant change in the values of lateral deflection for the variation of the weighing coefficients. And lastly it is observed that the sway values are still higher than those obtained from finite element analysis. This suggests the fact that the modification of ACI effective moment of inertia method by changing the weighing factors of $I_e$ is not fruitful and hence any effort in modifying the equation 6.2 is not going to take the sway values of the structure closer to the finite element solution.

## 6.4 STUDY OF EFFECTIVE MOMENT OF INERTIA CALCULATION

The second area of study is the power $\alpha$ of the ratio $M_{cr}/M_a$ in equation 6.1. Branson proposed equation 2.4 from which the generalized equation 6.1 is obtained. This equation calculates the effective moment of inertia of a member section. The equation clearly has two parts; one being the contribution of $I_g$, the gross moment of inertia of the section. The other one is the contribution of the cracked moment of inertia of the section $I_{cr}$.

Now the contribution of $I_g$ is obtained by multiplying it with a factor $(M_{cr}/M_a)^3$. The cracking moment $M_{cr}$ of a section is always smaller than the actual moment, $M_a$ for all practical cases. Hence this factor $M_{cr}/M_a$ becomes less than one most of the times. Now raising this factor to a power of 3 further reduces the over all value of the factor. This in turn means that the contribution of the gross moment of inertia $I_g$ is reduced further. Hence it is deduced that the effective moment of inertia has contributions of $I_{cr}$ larger than $I_g$. Now the aim at this point is to modify this situation so that the sway pattern of the structure becomes more representatives of the FE solutions. In this respect initiatives would be taken to test the change this power $\alpha$ to different values other than 3 and hence modify the contribution of $I_g$ towards $I_e$ calculation.

Questions may arise in this regard whether this approach is right or not. If the equation 6.1 is closely analyzed, it is observed again that $I_e$ provides a transition between the upper and the lower bounds $I_g$ and $I_{cr}$ respectively. In other words it is an interpolation between the well-defined limits of $I_g$ and $I_{cr}$. Then it can be written that,

$$I_{cr} < I_e < I_g$$

Hence during the current investigation the only thing that have to be kept in mind is that the value of $I_e$ calculated should lie within the value of $I_g$ and $I_{cr}$. If this is ensured then everything would be within the rules.

The third and final area of study is the calculation of $I_g$. As it was outlined in chapter 2, $I_g$ is calculated as $bh^3/12$. This considers the whole of the concrete neglecting any effect of reinforcements on $I_g$. Following common notion this should be modified to include the effect of both tension and compression steel. It has been just discussed that the contribution of $I_g$ must be increased to improve the sway values. One way to do that can be to increase the value of $I_g$ itself. This however can only be done if the contribution of the reinforcement towards $I_g$ calculation is considered. Some modifications, therefore, have been applied to the calculation of gross moment of inertia of the total uncracked section. As discussed earlier this modification of $I_g$ was incorporated in the investigation of variation of $\alpha$.

With the methodology just discussed in mind, the RC portal structure was analyzed with the experimental change in the power of $M_{cr}/M_a$. Remembering the parametric study, the sway values for different variation of the parameters have been calculated. But this time the sway was determined a number of times using various power of the multiplying factor of $I_g$. The lateral deflection values thus obtained was then plotted against the values obtained from finite element analysis.

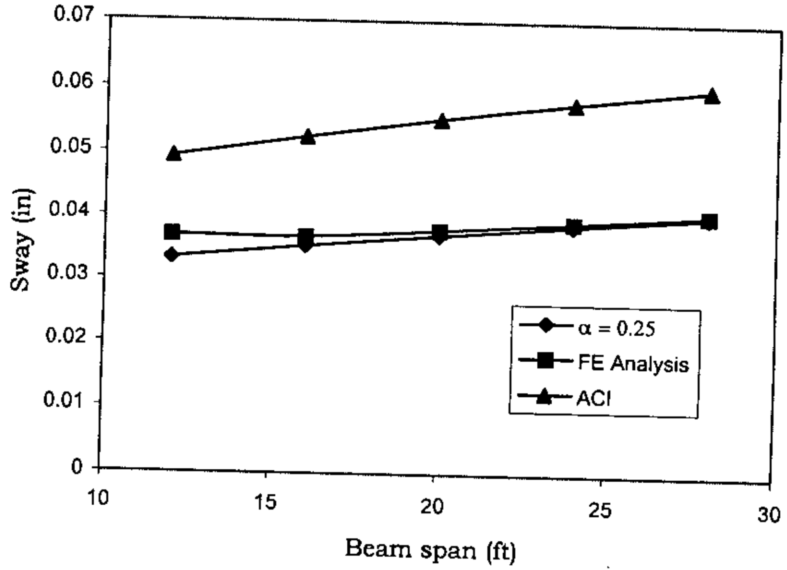



**Fig. 6.4: Sway vs. beam spans (ft) for values of α**

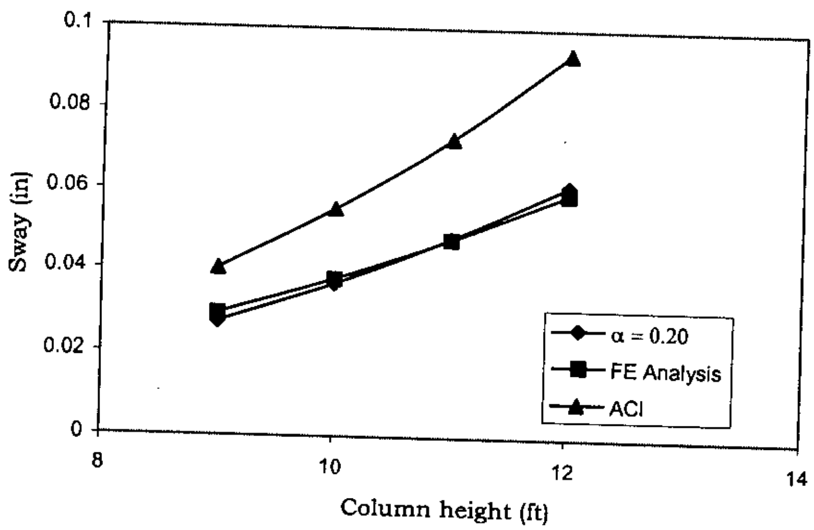



**Fig. 6.5: Sway vs. column heights for values of α**

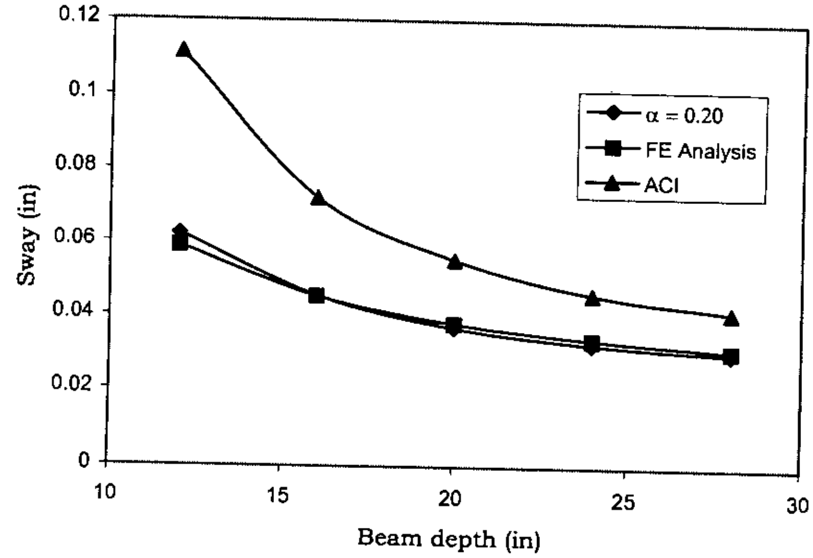



**Fig. 6.6: Sway vs. beam depths for values of α**

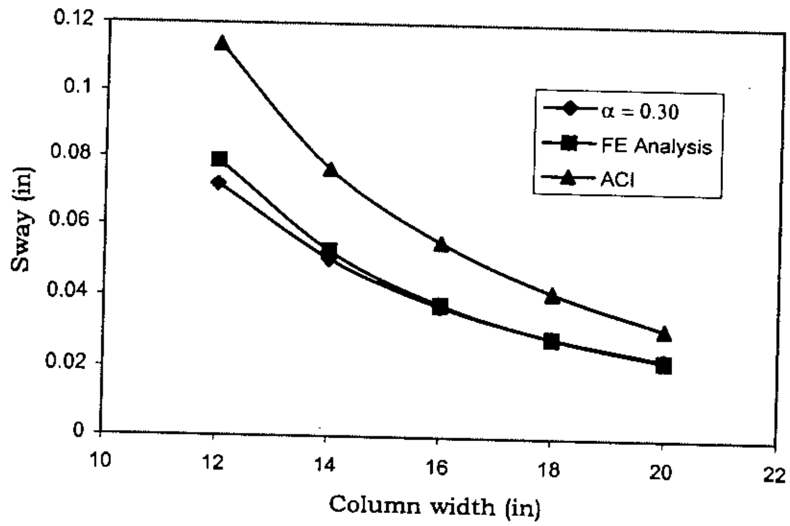



**Fig. 6.7: Sway vs. column widths for values of α**

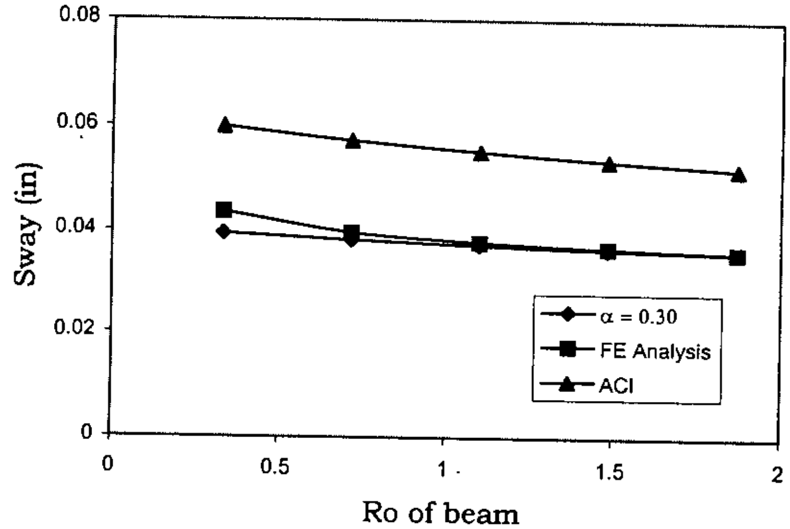



**Fig. 6.8: Sway vs. ρ of beam for values of α**

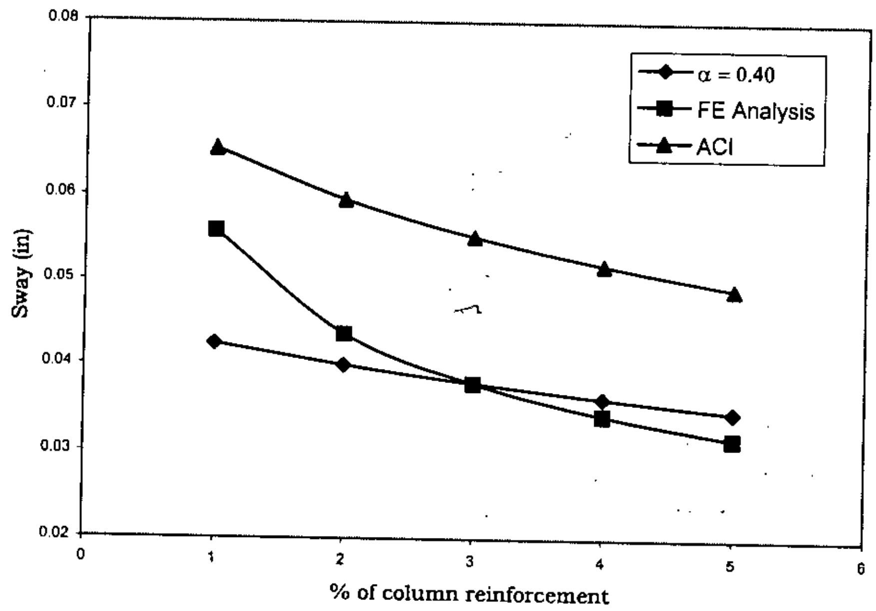



**Fig. 6.9: Sway vs. % of column reinforcement for values of α**

Fig. 6.4, 6.5, 6.6, 6.7, 6.8 and 6.9 illustrates the effect on the sway, calculated by ACI technique, due to the modification of the power of the factor $M_{cr}/M_a$, $\alpha$ and the modification of the $I_g$ calculation procedure. In all of the cases it is seen that modifying the power from 3 to some other value takes the values of sway near to the values obtained by finite element analysis. This indeed is a promising feature of the study. It should be mentioned here that these power values were obtained by extensive trial and error procedure and the curve were plotted for only those values of the power that yielded sways very near to the finite element solution.

It should also to be noted that for all the graphs from fig. 6.4 to 6.9 the values of $\alpha$ were found to be within a range of 0.20 to 0.40. This is another promising feature in a sense that, since these values are close enough, some median value may be applied in all these cases with some tolerance. Thus it can concluded that modifying the power of $M_{cr}/M_a$ i.e. $\alpha$ of equation 6.1 does provide with an estimation of the sway closer to the FE solutions using guidelines proposed by ACI.

## 6.5 SUGGESTED MODIFICATIONS TO THE ACI METHOD

Before the suggested modifications are proposed some facts must be reiterated. It has been discussed earlier that effective moment of inertia $I_e$ is in fact an interpolation between the values of gross moment of inertia $I_g$ and cracked moment of inertia of the section $I_{cr}$. It provides a transition between the upper and lower bounds of $I_g$ and $I_{cr}$ respectively. Thus it can be written safely that $I_{cr} < I_e < I_g$. Now it is known that the value of $I_g$ is much greater than the value of $I_{cr}$. Therefore when $I_e$ approaches $I_g$ then its value would be larger and vice versa. It is the general understanding that frames with stiff member sections are likely to give smaller sway values. Which in other terms means that if the moments of inertia of sections are increased, lateral deflection of the structure would be reduced. Hence to minimize sway the aim would be to have sections with larger moments of inertia. In this respect the higher the values of $I_e$ the lower would be the sway. Therefore it is beneficial towards sway calculation, to have $I_e$ values closer to $I_g$.

It was seen earlier that the first modification did not actually yield any significant change in sway values of the structure. Even though various combinations of weighing factors were applied to equation 6.2 the result was not up to the mark. Generally the value of $I_e$ should increase with lower values of $\omega$, because in this case the contribution of midspan $I_e$ is increased and at midspan $I_e$ values are expected to be larger. But no reflection of this fact was observed in the study. This may be attributed to the cause that the $I_e$ value for midspan itself has a lower magnitude and improving its contribution does not effectively increase the value of total effective moment of inertia. If by some method the $I_e$ of the midspan could have been increased smaller sway values would have been obtained closer to the finite element solution for relatively lower values of the weighing factor $\omega$ of equation 6.2. Hence it is concluded that equation 6.2 should be used as it is.

The second sets of graphs (fig. 6.4 through 6.9) of the analysis yielded better results. In fact sway values were obtained which are very close the values obtained from the finite element analysis. This was possible for two modifications described as (b) and (c) in section 6.2. Comments would be made on them in the following paragraphs.

During the calculation of $I_e$, $I_g$ was determined incorporating the effect of reinforcements. The original recommendation of $I_g$ calculation by ACI is given by equation 6.5, while the modified calculation in this case was adopted following the equation 6.6.

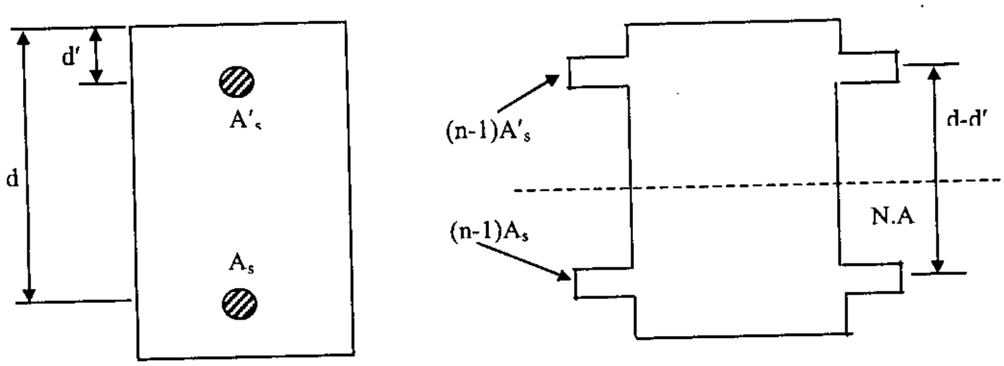


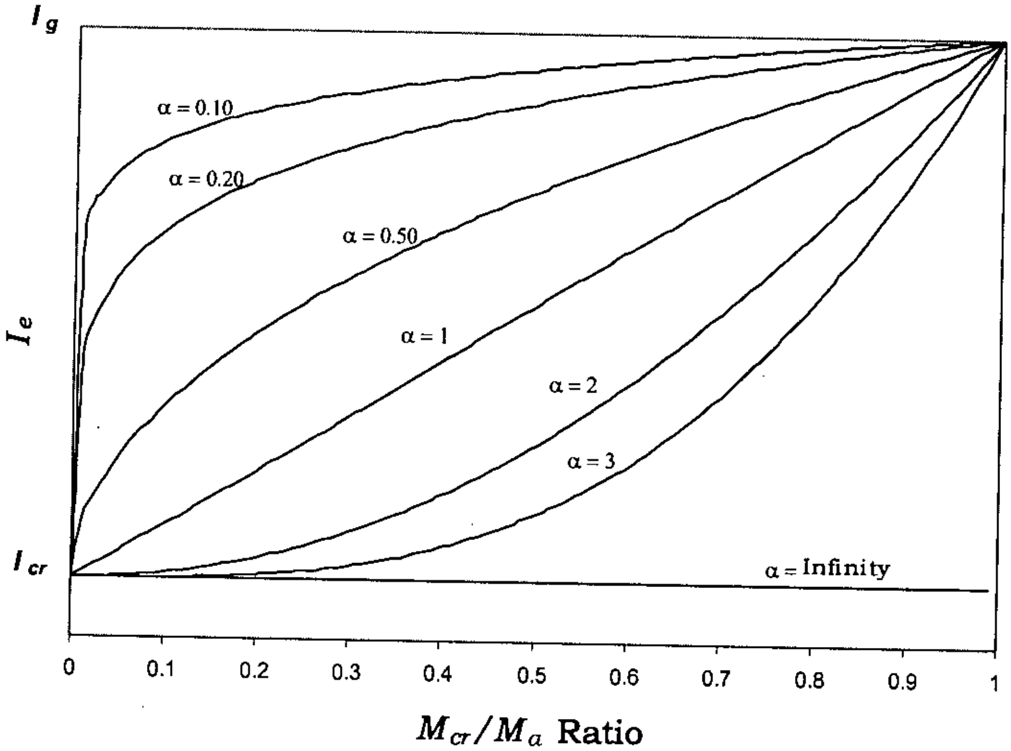

**Fig. 6.10: Uncracked transformed section**

$$I_g = \frac{bh^3}{12} \quad 6.5$$

$$I_g = \frac{bh^3}{12} + (n-1)A_s\left(\frac{d-d'}{2}\right)^2 + (n-1)A_s'\left(\frac{d-d'}{2}\right)^2 \quad 6.6$$

Equation 6.6 clearly shows the effect of both tension and compression reinforcements towards $I_g$ calculation. It is obvious from the equation that the $I_g$ value obtained from equation 6.6 would be larger than that obtained from equation 6.5. Hence using equation 6.6, There is a better chance of obtaining larger values of $I_e$. This in turn means that using equation 6.6 would make the sections stiffer and hence help towards reducing the sway values more towards the values obtained from finite element analysis. Actually this fact is observed to be in effect in the graphs 6.4, 6.5, 6.6, 6.7, 6.8 and 6.9. In summary, the moment of inertia of uncracked transformed section is suggested as $I_g$.

The effect of the power of $M_{cr}/M_a$ in equation 6.1 $\alpha$ is more prominent on the sway estimation of frame. As discussed previously the closer the value of $I_e$ towards the value of $I_g$ the lower would be the value of lateral deflection. Now using equation 6.1 one can increase the value of $I_e$ by modifying $\alpha$. In this connection fig. 6.11 can be observed. It shows a variation of $I_e$ values with the variation of the ratio $M_{cr}/M_a$ for different values of $\alpha$. It is evident from this graph that when the value of $\alpha$ increases the value of $I_e$ is closer to the cracking moment of inertia $I_{cr}$. For an infinite value of $\alpha$ the values of $I_e$ matches the exact value of $I_{cr}$. On the other hand if the value of $\alpha$ is reduced the value of $I_e$ edges towards the value of $I_g$. Now to minimize the sway towards the FEM solutions $I_e$ needs to be close enough to $I_g$. Hence it follows that if lower values of $\alpha$ are used the values of effective moment of inertia would larger and the lateral deflection pattern obtained would be lower and close to the values obtained from finite element analysis. Fig. 6.4, 6.5, 6.6, 6.7, 6.8 and 6.9 show remarkable proof of this fact. In all the cases sway patterns closely matching the finite element solution were obtained with lower values of $\alpha$. In fact the values of $\alpha$ that produces the desired effect are all within a small range of 0.20 and 0.40. This is to be noted here that the value of $\alpha$ is 3 according to the ACI guidelines and is far more than the values obtained from the investigations. The values of $\alpha$ obtained from the study also conforms to fig. 6.11. Since these values are lower than 1 they




tend to give $I_e$ values close to $I_g$ and hence help to reduce the sway.

**Fig. 6.10: $I_e$ vs. $M_{cr}/M_a$ ratio for values of $\alpha$**

As for the values of $\alpha$ it has already been discussed that they range between 0.20 and 0.40. Now in the current parametric study different values $\alpha$ were obtained for different parameters that produce the closest possible match of the finite element solutions. Therefore, it can be suggested that the value of $\alpha$ can be a function of a particular parameter and hence may be treated as a variable. But this would hinder the efforts in finding out a simpler modification to the ACI guideline. Hence $\alpha$ would not be treated as a variable for different parameters of the structure. The way left now is to adopt a median value of $\alpha$ within the range 0.20 - 0.40. The value of $\alpha$, therefore, has been taken to be 0.30. It can be concluded that this value of $\alpha$ can be safely used to estimate the sway of a structure with minimum tolerance to the finite element solutions. Also it should be mentioned that this value is strictly for a simple portal frame.

The final propositions of the study, therefore, are as follows.

1. The gross moment of inertia of the total uncracked section should be calculated incorporating the effects of reinforcement. For a doubly reinforced rectangular section the value of $I_g$ should be calculated as follows,

$$I_g = \frac{bh^3}{12} + (n-1)A_s\left(\frac{d-d'}{2}\right)^2 + (n-1)A_s'\left(\frac{d-d'}{2}\right)^2$$

2. It has been established from the study that the ACI formula for $I_e$ calculation (equation 6.1) may not be fully applicable for calculation of sway of RC frames. This formula was proposed by Branson (1977, 1982 and 1985) for calculating deflection of simple and continuous beams and probably is still appropriate for beams. This may be due to the fact that for beams the pattern of bending moment under vertical load is such that the applied moments are high both at supports and at mid span. Which means that sections are cracked at these locations. This implies that contribution of $I_{cr}$ towards $I_e$ shall be more than $I_g$. This is exactly the fact that Branson's formula utilizes which can be seen from fig. 6.10 for $\alpha = 3$. The transition of $I_e$ from $I_{cr}$ to $I_g$ is such that for most part $I_e$ remains close to $I_{cr}$. The pattern of bending moment in a frame under lateral load is quite different from that of beam, having minimum amount of moment at mid span and opposite moments at ends. Therefore contribution of $I_{cr}$ towards $I_e$ may not be as high as in the cases of beams under gravity load. Therefore the Branson's formula may be modified to reduce the contribution of $I_{cr}$ and increase the contribution of $I_g$. This can be done by reducing the value of $\alpha$ from 3 to a smaller value. For a simple portal subjected to lateral loads only as studied in this thesis, this may be as low as 0.3. However the value of $\alpha$ to be used for the determination of sway of RC frames under combined action of lateral and gravity loads still requires further study which may not be decided on the basis of limited study presented in this thesis.
3. Regarding the average of $I_e$, ACI guidelines (equation 2.5, 2.6 and 2.7) may be continued to use.

## 6.6 PERFORMANCE OF THE PROPOSAL

Any scientific proposition requires performance checking. The suggested modifications of the ACI guideline hence must be tested to make it acceptable. For this purpose the modifications have been applied in the analysis of structures other than the simple portal frame. The proposal was tested for the structure shown in fig. 6.12, using the value of $\alpha$ = 0.3 as suggested by the current investigation. This structure has beam and column dimensions same as the initial values of the simple portal structure analyzed in chapter 5. The structure was subjected to the same lateral load of 5 kips at the top. The structure was analyzed using finite element modeling first. The modeling technique was similar to the one adopted for the simple portal frame. The lateral deflection was then estimated. The same frame was analyzed using the guidelines of ACI committee report 635R-95 with due modifications proposed in this chapter. For the FE analysis the sway was obtained to be 0.207 inch.

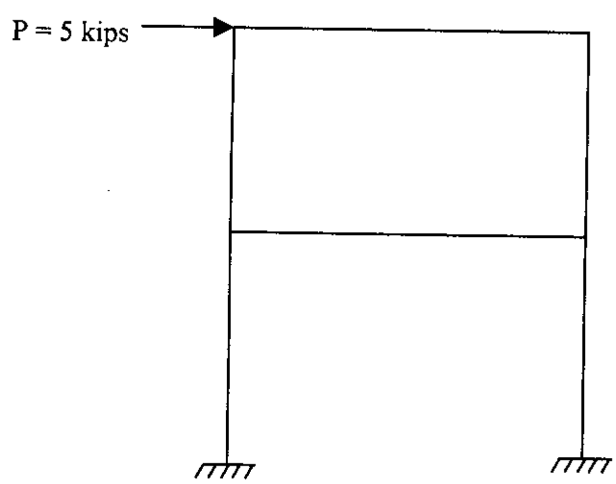



**Fig. 6.12: Skeletal structure of the test frame**

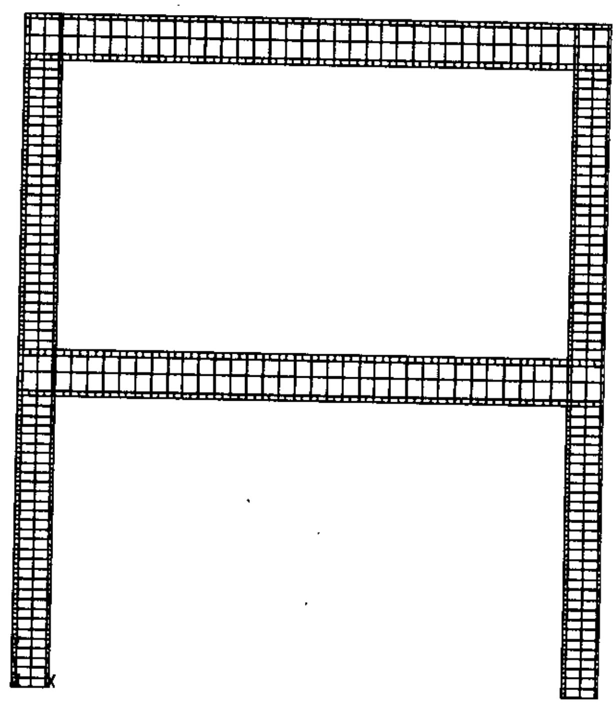


**Fig.6.13: Finite element model mesh of 2 story portal structure**

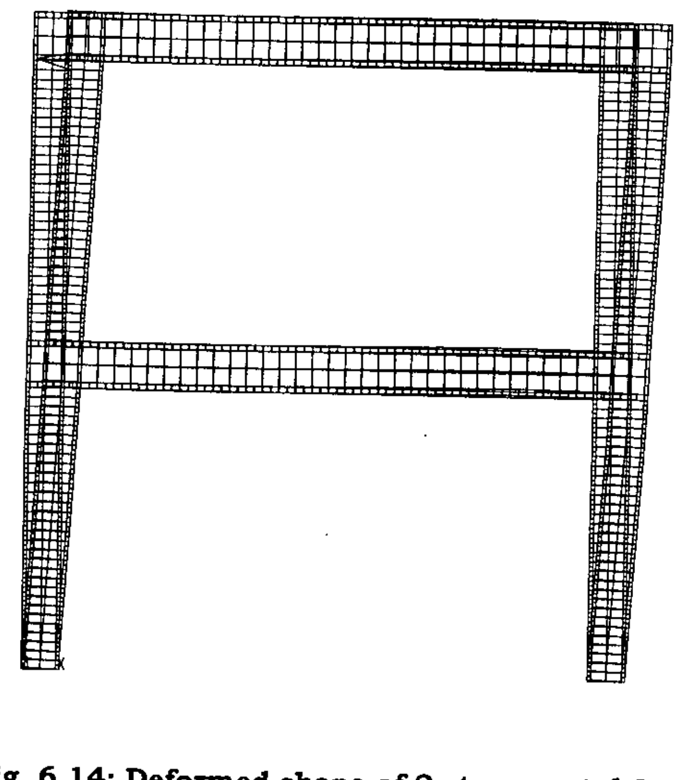


**Fig. 6.14: Deformed shape of 2 story portal frame**

When the model was analyzed using the original ACI guideline the sway value calculated was 0.335 inch. However the analysis of the structure using the ACI guideline along with proposed modification resulted in a lateral deflection value of 0.223 inch.

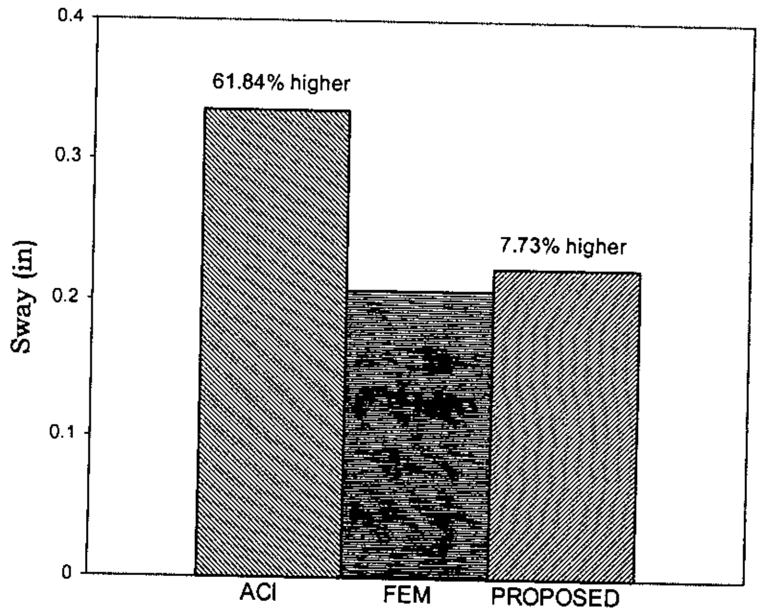



**Fig. 6.15: Comparative analysis of results from different methods**

## Chapter 7

# CONCLUSION

### 7.1 GENERAL

The purpose of this thesis was to devise a rational guideline for the estimation of sway of RC beam and column structures. The guideline should be one that can be easily employed and overcomes any overestimation of sway thereby reducing the section proportions and ensuring economy in design. In this regard the guidelines proposed by the ACI committee report 435R-95 was adopted as the basis. The guidelines being laid down for beams, an approach dictating its adaptability to portal frame structures was discussed. To check the performance of the approach a simple portal frame was selected for extensive analysis. The portal frame was first analyzed using the guidelines from the ACI code. A verification of this analysis was performed by comparing the results obtained with those of a finite element analysis of the same structure. The FE analysis of the RC structure was performed using special discretization technique having features like separate elements for different components of concrete i.e. concrete and steel. Special elements were used in the modeling for simulating bond-slip mechanism at the interface of concrete and reinforcement. The validity of using such arrangements, however, was checked first by comparing the analyses of a RC beam and column joint, using both conventional and finite element techniques. Having established the fact, that some rotational characteristics do exist within RC joints the discretization technique was adopted for the portal frame analysis. For better understanding of the difference between the sway patterns of FE analysis and ACI guidelines a detailed parametric study was performed. Depending on the difference of the sway values calculated, efforts were made to investigate proper modifications to the technique outlined by ACI, so that this method estimates the sway closer to the finite element solutions. Upon the completion of this investigation appropriate fields were identified within the ACI analysis, which requires modification towards refined estimation of sway values. A systematic modification of those fields was then proposed. Finally the performance of the suggested modifications were investigated by applying them to another structure other than the simple portal frame. The structure for performance checking was also analyzed using the original ACI approach and finite element method. Finally a comparison of the sway values from these three analyses was discussed.

## 7.2 CONCLUSIONS

From the investigations carried out the following important deductions can be made.

- The method for calculating RC beam deflections, as provided in ACI committee report 435R-95, may be adapted to estimate the sway value of RC frame structures.
- The lateral deflections of RC portal frame varies considerably when analysed using the guidelines of ACI 435R-95 and finite element modeling.
- The sway values obtained from finite element solutions are generally less than those obtained from the conventional ACI technique.
- The ACI effective moment of inertia method of analysis has three distinct areas, which can be modified to improve the sway results.
- The weighing coefficients used in the ACI method for over all effective moment of inertia calculation has less effect on the lateral deflection calculations. For variations in the weighing factor the sway values are reduced only a small amount and the sway patterns thus obtained are almost parallel.
- Making the effective moment of inertia close to the gross moment of inertia of the sections, sway of the structure can predicted more reasonably.
- The effective moment of inertia of the structure is taken closer towards the gross moment of inertia by reducing the power of the ratio of cracking

moment and actual moment of the structure. Hence by reducing this power sway values can be estimated which are more representative of real situations.

- Moment of inertia f uncracked transformed section may be used as the gross moment of inertia.
- With proper modifications, the ACI effective moment of inertia method may be employed to estimate the sway of a simple RC portal frame with an acceptable level of accuracy that is unattainable using conventional analysis.

## 7.3 RECOMMENDATION FOR FUTURE INVESTIGATIONS

Following the objective of this thesis, the ACI effective moment of inertia method with some modifications, has been suggested for sway estimation of RC portal structure. However additional investigation may be performed to make this method an effective tools for regular use. Here are some points that demand future investigation in this regard.

- The proposed modifications of the ACI technique must be subjected to more extensive testing to prove its acceptability. It has to be applied to different portal frame arrangements and its performance must be checked against finite element models.
- The validity of the proposed modifications should be checked against real life models.
- Parametric study performed in this thesis may be continued for structures having more than a single portal arrangement.
- To handle the iteration process involved in the method, computer software package may be developed.

# APPENDIX

## ANSYS Parameters Definition

| | |
|---|---|
| **COLSIZE** | Column depth |
| **BEAMHT** | Beam depth |
| **COLHT** | Column height |
| **BEAMSPAN** | Beam span |
| **BRDIA** | Reinforcing bar diameter for beam |
| **CRDIA** | Reinforcing bar diameter for column |
| **FPRIMEC** | Concrete compressive strength |
| **FY** | Ultimate strength of steel |
| **ROMAX** | Maximum steel ratio |
| **ROMIN** | Minimum steel ratio |
| **ROAVG** | Average steel ratio |
| **COLRPERC** | Percentage of column reinforcement |
| **FT** | Concrete tensile strength |

# ANSYS Script File for Parameters

```
/NOPR

!*** Set Column Size Here ***
!*SET,COLSIZE,12
!*SET,COLSIZE,14
*SET,COLSIZE,16
!*SET,COLSIZE,18
!*SET,COLSIZE,20
!*****************************

!*** Set Beam Height Here ***
!*SET,BEAMHT,12
!*SET,BEAMHT,16
*SET,BEAMHT,20
!*SET,BEAMHT,24
!*SET,BEAMHT,28
!*****************************


!*** Set Column Height Here ***
!*SET,COLHT,108 - BEAMHT/2
*SET,COLHT,120 - BEAMHT/2
!*SET,COLHT,132 - BEAMHT/2
!*SET,COLHT,144 - BEAMHT/2
!*****************************


!*** Set Beam Span Here ***
!*SET,BEAMSPAN,144 - COLHT
!*SET,BEAMSPAN,192 - COLHT
*SET,BEAMSPAN,240 - COLHT
!*SET,BEAMSPAN,288 - COLHT
!*SET,BEAMSPAN,336 - COLHT
!*****************************


!*** Set Bar Dia For Beam Here ***
*SET,BRDIA,6/8
!*****************************


!*** Set Bar Dia For Beam Here ***
*SET,CRDIA,6/8
!*****************************


!*** Set FPrimeC Here ***
*SET,FPRIMEC,3500
!*****************************

!*** Set Fy Here ***
*SET,FY,60000
!*****************************


*SET,ROMAX,0.75*0.85*0.85*FPRIMEC/FY*87/(87+FY/1000)
*SET,ROMIN,200/FY
*SET,ROAVG,(ROMAX+ROMIN)/2
```

```
!*** Set Ro Here ***
!*SET,RO,ROMIN
!*SET,RO,(ROMIN+ROAVG)/2
*SET,RO,ROAVG
!*SET,RO,(ROAVG+ROMAX)/2
!*SET,RO,ROMAX
!****************************


!*** Set Column Reinforcement Percentage Here ***
!*SET,COLRPERC,1
!*SET,COLRPERC,2
*SET,COLRPERC,3
!*SET,COLRPERC,4
!*SET,COLRPERC,5
!****************************


*SET,D,BEAMHT - 2.5
*SET,DPRIME,2.5


!*** Set Ft Value Here ***
!*SET,FT,3*SQRT(FPRIMEC)
*SET,FT,4*SQRT(FPRIMEC)
!*SET,FT,5*SQRT(FPRIMEC)
!****************************



*SET,BMRAREA,RO*BEAMHT*12
*SET,NBBEAM,BMRAREA/(3.141*BRDIA*BRDIA/4)

*SET,COLRAREA,(COLSIZE*12)*COLRPERC/100/3
*SET,NBCOL,COLRAREA/(3.141*CRDIA*CRDIA/4)

*SET,COLDIV,30



*SET,CONCRETE,1
*SET,STEEL,2

*SET,PL82c,1
*SET,PL42c,2
*SET,REINFc,3
*SET,PL82b,4
*SET,PL42b,5
*SET,REINFb,6
*SET,BEAMSPRG,7
*SET,COLSPRG,8


*SET,_RETURN ,  .0000000000000
*SET,_STATUS ,  1.000000000000
*SET,_ZX     ,'            '
/GO
```

# ANSYS Script File for Exterior Joint Analysis

```
/prep7
parres,,param
et,,plane82,,,3,,2,3
et,,plane42,,,3,,2,3
et,,link1
et,,link1
et,,link1
et,,combin14,,1
mp,ex,1,3e6
mp,nuxy,1,.17
mp,ex,2,30e6
mp,nuxy,2,.3
/units,bin
r,1,10
r,2,10
r,3,barnos*barsize*barsize*22/64/7/4
r,4,tbararea
r,5,cbararea
r,6,94758.3*barsize*barnos*2.5
r,7,94758.3*barsize*barnos*5
r,8,94758.3*barsize*barnos*7.5
r,9,94758.3*barsize*barnos*10
r,10,mkt*2.5
r,11,mkt*5
r,12,mkt*7.5
r,13,mkt*10
/pnum,node,1
n,1,,-15*0.5
n,2,,-15*0.5
ngen,2,2,1,,,,(15-2.5)*0.5
ngen,2,3,1,2,,,(15-2.5)
ngen,2,5,1,,,,15
ngen,2,6,1,,,1.5
ngen,2,5,3,,,1.5
ngen,2,5,4,,,1.5
ngen,2,4,6,,,1.5
ngen,2,10,1,2,,5
ngen,2,9,4,5,,5
ngen,2,9,6,,,5
ngen,2,9,7,10,,7
ngen,2,19,1,6,,10
ngen,2,25,1,,,15
ngen,2,23,4,,,15
ngen,2,22,6,,,15
ngen,2,9,20,25,,10
ngen,12,9,26,34,,10
ngen,2,128,6,,,,6
ngen,2,120,15,,,,6
ngen,2,111,25,,,,6
ngen,2,3,134,,,,6
ngen,2,1,137,,,1.5
ngen,2,2,137,,,5
ngen,2,3,137,138,,8.5
ngen,10,8,134,141,,,12
nsym,y,80,134,213
TYPE,1,
MAT,1,
REAL,1,
```

```
E,20,29,32,23,26,31,27,22
TYPE,2,
MAT,1,
REAL,2,
E,25,23,27,28
E,28,27,32,34
TYPE,3,
MAT,2,
REAL,3,
E,21,30
TYPE,4,
MAT,2,
REAL,4,
E,24,33
EGEN,12,9,1,5
TYPE,1,
MAT,1,
REAL,1,
E,6,15,139,137,10,135,138,134
E,15,25,141,139,19,136,140,135
TYPE,5,
MAT,2,
REAL,5,
E,10,138
E,15,139
E,19,140
TYPE,1,
MAT,1,
REAL,1,
E,137,139,147,145,138,143,146,142
E,139,141,149,147,140,144,148,143
TYPE,5,
MAT,2,
REAL,5,
E,138,146
E,139,147
E,140,148
EGEN,9,8,66,70
TYPE,1,
MAT,1,
REAL,1,
E,217,219,11,1,218,215,7,214
E,219,221,20,11,220,216,16,215
E,225,227,219,217,226,223,218,222
E,227,229,221,219,228,224,220,223
TYPE,5,
MAT,2,
REAL,5,
E,7,218
E,11,219
E,16,220
E,218,226
E,219,227
E,220,228
FLST,4,5,2,ORDE,4
FITEM,4,113
FITEM,4,-114
FITEM,4,118
FITEM,4,-120
EGEN,9,8,P51X, , , , , , ,
TYPE,1,
```

```
MAT,1,
REAL,1,
E,7,16,18,9,11,17,13,8
TYPE,2,
MAT,1,
REAL,2,
E,1,7,8,3
E,3,8,9,4
E,4,13,15,6
E,13,23,25,15
E,17,22,23,18
E,16,20,22,17
TYPE,3,
MAT,2,
REAL,3,
E,21,12
E,12,2
TYPE,4,
MAT,2,
REAL,4,
E,24,14
E,14,5
TYPE,5,
MAT,2,
REAL,5,
E,10,9
E,9,8
E,8,7
E,15,13
E,13,11
E,19,18
E,18,17
E,17,16
D,209,ALL, , ,213
D,289,ALL, , ,293
TYPE,6,
MAT,2,
REAL,6,
E,2,1
real,7
E,12,11
real,8
E,21,20
real,9
E,30,29
EGEN,11,9,183
real,7
E,129,128
real,10
E,5,4
real,11
E,14,13
real,12
E,24,23
real,13
E,33,32
EGEN,11,9,198
real,11
E,132,131
CPINTF,UY
```

# ANSYS Script File for Simple Portal Frame Analysis

```
parres,,param.txt

/prep7

!This ensures that node numbering is on

!/pnum,node,1

!This starts drawing the nodes of
!left column from bottom

!***************************
n,1,0
n,2,1.5
n,3,1.5
n,4,(colsize-3)/4+1.5
n,5,colsize/2
n,6,colsize/2
n,7,colsize-1.5-(colsize-3)/4
n,8,colsize-1.5
n,9,colsize-1.5
n,10,colsize
n,11,0,colht/60
n,12,1.5,colht/60
n,13,colsize/2,colht/60
n,14,colsize-1.5,colht/60
n,15,colsize,colht/60
ngen,30,15,1,15,,0,colht/30
ngen,2,450,1,10,,0,colht
!****************************


!Following block gives extra 7 nodes at column top

!****************************
n,461,0,colht
n,462,1.5,colht
n,463,(colsize-3)/4+1.5,colht
n,464,colsize/2,colht
n,465,colsize-1.5-(colsize-3)/4,colht
n,466,colsize-1.5,colht
n,467,colsize,colht
!****************************


!The folloeing block gives the joint atop the column

!****************************
ngen,2,467,1,10,,0,colht + beamht - d
ngen,2,467,11,15,,0,colht + (d - dprime) / 4 + beamht - d -
(colht/60)  !** 123.75 becuse node 11-15 are 2 inch higher above base

ngen,2,15,468,482,,0,(d - dprime) /2
ngen,2,15,483,492,,0,(d - dprime) /2

ngen,2,47,461,467,,0,beamht
!****************************
```

```
!Drawing the other column

!****************************
ngen,2,514,1,514,,beamspan + colsize,0
!****************************


!Now drawing the beam
!First line

!****************************
n,,colsize,colht + beamht
n,,colsize,colht + beamht - dprime
n,,colsize,colht + beamht - dprime
n,,colsize,colht + beamht - dprime - (d - dprime) / 4
n,,colsize,colht + beamht - dprime - (d - dprime) / 2
n,,colsize,colht + beamht - dprime - (d - dprime) / 2
n,,colsize,colht + beamht / 2 - (d - dprime) / 4
n,,colsize,colht + beamht - d
n,,colsize,colht + beamht - d
n,,colsize,colht
!****************************


!Second line

!****************************
n,,colsize + beamspan/90 ,colht + beamht
n,,colsize + beamspan/90 ,colht + beamht - dprime
n,,colsize + beamspan/90 ,colht + beamht - dprime - (d - dprime) / 2
n,,colsize + beamspan/90 ,colht + beamht - d
n,,colsize + beamspan/90 ,colht
!****************************


!Generating intermeiate lines

!****************************
ngen,45,15,1029,1043,,beamspan/45
!****************************

!Last line

!****************************
ngen,2,675,1029,1038,,beamspan
!****************************


!Creates extra nodes at the joint for flexural reinforcement
!****************************
n,,1.5,colht + beamht - d
n,,colsize/2,colht + beamht - d
n,,colsize - 1.5,colht + beamht - d

ngen,3,3,1714,1716,,,(d - dprime)/2
!****************************

!Copying to the other joint
!****************************
ngen,2,9,1714,1722,,colsize + beamspan
```

```
!****************************

!!!!!! END OF NODE GENERATION !!!!!!

!Units definition

!****************************
/units,bin
!****************************

!!!!!! START OF ELEMENT, MATERIAL REALCONSTANT DEFINITION !!!!!!

!Defining Concrete Property

!****************************
mp,ex,concrete,3e6
mp,nuxy,concrete,.17
!****************************

!Defining Steel Property

!****************************
mp,ex,steel,30e6
mp,nuxy,steel,.3
!****************************

!Definition of element

!****************************
et,pl82c,plane82
keyopt,1,3,3
keyopt,1,5,0
keyopt,1,6,0
et,pl42c,plane42,,,3,,2,3
et,reinfc,link1
et,pl82b,plane82,,,3,,2,3
et,pl42b,plane42,,,3,,2,3
et,reinfb,link1
et,beamsprg,combin14,0,1          !spring with UX dof, used in beam
et,colsprg,combin14,0,2             !spring with UY dof, used in
column
!****************************

!Definition of Real Constants

!****************************
!Column element thickness plane82
!width of column
r,1,12

!Column element thickness plane42
!width of column
```

```
r,2,12

!Column reinforcement area divided by 3 link
!width of reinf. in column
r,3,COLRAREA

!Beam element thickness plane82
!width of beam
r,4,12

!Beam element thickness plane42
!width of column
r,5,12

!Beam element reinforcement link (actual area of either the top or
bottom reinf.)
!width of reinf
r,6,BMRAREA

!K value for spring used in beam
!value of k of the spring
r,7,2526.888*BRDIA*(BEAMSPAN/45)*NBBEAM*FT

!K value for spring used in column
!value of k of the spring
r,8,2526.888*CRDIA*(COLHT/30)*NBCOL*FT

r,9,0.01
!****************************


!!!!!! END OF ELEMENT, MATERIAL REALCONSTANT DEFINITION !!!!!!


!!!!!! START OF ELEMENT GENERAITION !!!!!!


!Left Column

!Generating Concrete Elements

!****************************
type,pl82c
mat,concrete
real,pl82c
!****************************

!First 8 noded elements
!****************************
e,2,5,20,17,4,13,19,12
e,5,8,23,20,7,14,22,13
!****************************


!****************************
type,pl42c
mat,concrete
real,pl42c
!****************************
```

```
!Now 4 noded elements
!****************************
e,1,2,12,11
e,8,10,15,14
e,11,12,17,16
e,14,15,25,23
!****************************


!****************************
type,reinfc
mat,steel
real,reinfc
!****************************

!Now generating the reinforcement
!****************************
e,3,18
e,6,21
e,9,24
!****************************


!****************************
type,colsprg
mat,steel
real,colsprg
!****************************

!Now generating the spring
!****************************
e,2,3
e,5,6
e,8,9
!****************************


!Copying the elements generated so far
!****************************
egen,30,15,1,12
!****************************


!****************************
type,colsprg
mat,steel
real,colsprg
!****************************

!The last layer of spring in the left column
!****************************
e,452,453
e,455,456
e,458,459
!****************************

!Generating the left joint

!****************************
type,pl42c
mat,concrete
```

```
real,pl42c
!****************************

!4noded elements
!****************************
e,461,462,469,468
e,462,463,471,469
e,463,464,472,471
e,464,465,474,472
e,465,466,475,474
e,466,467,477,475

e,498,499,509,508
e,499,501,510,509
e,501,502,511,510
e,502,504,512,511
e,504,505,513,512
e,505,507,514,513

e,468,469,479,478
e,475,477,482,481
e,478,479,484,483
e,481,482,492,490
e,483,484,494,493
e,490,492,497,496
e,493,494,499,498
e,496,497,507,505

!****************************


!****************************
type,pl82c
mat,concrete
real,pl82c
!****************************

!8noded elements
!****************************
e,469,472,487,484,471,480,486,479
e,472,475,490,487,474,481,489,480
e,484,487,502,499,486,495,501,494
e,487,490,505,502,489,496,504,495
!****************************


!****************************
type,reinfc
mat,steel
real,reinfc
!****************************

!Link elements
!****************************
e,453,470
e,456,473
e,459,476

e,470,485
e,473,488
e,476,491
```

```
e,485,500
e,488,503
e,491,506
!*****************************


!*****************************
type,colsprg
mat,steel
real,colsprg
!*****************************

!Spring elements
!*****************************
e,469,470
e,472,473
e,475,476

e,484,485
e,487,488
e,490,491

e,499,500
e,502,503
e,505,506
!*****************************


!***** END OF LEFT COLUMN *****


!***** GENERATING RIGHT COLUMN ******

!*****************************
egen,2,514,1,405
!*****************************

!***** END OF RIGHT COLUMN *****


!***** GENERATING THE BEAM ******


!*****************************
type,pl82b
mat,concrete
real,pl82b
!*****************************

!First 8 noded elements
!*****************************
e,1030,1033,1048,1045,1032,1041;1047,1040
e,1033,1036,1051,1048,1035,1042,1050,1041
!*****************************


!*****************************
type,pl42b
```

```
mat,concrete
real,pl42b
!****************************

!Now 4noded elements
!****************************
e,1029,1030,1040,1039
e,1039,1040,1045,1044

e,1036,1038,1043,1042
e,1042,1043,1053,1051
!****************************


!****************************
type,reinfb
mat,steel
real,reinfb
!****************************

!Reinforcements
!****************************
e,1031,1046

real,9
e,1034,1049

real,reinfb
e,1037,1052
!****************************


!****************************
type,beamsprg
mat,steel
real,beamsprg
!****************************

!Springs
!****************************
e,1030,1031
e,1033,1034
e,1036,1037
!****************************


!Copying the elements generated so far
!*************************************
!egen,beamspan / 4,15,811,822
egen,45,15,811,822
!*************************************


!Generating the last layer of spring in the beam


!****************************
e,1705,1706
e,1708,1709
e,1711,1712
!****************************
```

```
!***** END OF BEAM GENERATION *****

!***** HORIZONTAL REINFORCEMENTS IN JOINTS *****

!****************************
type,beamsprg
mat,steel
real,beamsprg
!****************************

!Beam reinforcement spring on left joint
!****************************
e,469,1714
e,472,1715
e,475,1716

e,484,1717
e,487,1718
e,490,1719

e,499,1720
e,502,1721
e,505,1722
!****************************

!Beam reinforcement spring on right joint
!****************************
e,983,1723
e,986,1724
e,989,1725

e,998,1726
e,1001,1727
e,1004,1728

e,1013,1729
e,1016,1730
e,1019,1731
!****************************

!****************************
type,reinfb
mat,steel
real,reinfb
!****************************

!First Left Joint
!****************************
e,1031,1722
e,1034,1719
e,1037,1716

e,1722,1721
e,1719,1718
e,1716,1715

e,1721,1720
e,1718,1717
```

```
e,1715,1714
!****************************

!Then Right Joint
!****************************
e,1706,1729
e,1709,1726
e,1712,1723

e,1729,1730
e,1726,1727
e,1723,1724

e,1730,1731
e,1727,1728
e,1724,1725
!****************************


!***** END OF ELEMENTS IN BOTH JOINTS *****



!***** APPLYING COUPLING OF CO-INCIDENT NODES *****

!First for Columns
!****************************
cp,1,ux,3,2              !Left
cpsgen,2,3,,,,           !Left
cpsgen,2,6,,,,           !Left
cpsgen,2,514,,,,         !Right
cpsgen,2,517,,,,         !Right
cpsgen,2,520,,,,         !Right
cpsgen,31,15,1,6,,       !Copying

cpsgen,2,467,,,,
cpsgen,2,470,,,,
cpsgen,2,473,,,,
cpsgen,3,15,187,189,,

cpsgen,2,981,,,,
cpsgen,2,984,,,,
cpsgen,2,987,,,,
cpsgen,3,15,196,198,,
!****************************

!For Beam
!****************************
cp,205,uy,1030,1031
cpsgen,2,6,205,,,
cpsgen,46,15,205,206,,

!in left joint

cp,297,uy,499,1720
cp,298,uy,484,1717
cp,299,uy,469,1714

cp,300,uy,502,1721
cp,301,uy,487,1718
```

```
cp,302,uy,472,1715

cp,303,uy,505,1722
cp,304,uy,490,1719
cp,305,uy,475,1716

!in right joint

cp,306,uy,983,1723
cp,307,uy,986,1724
cp,308,uy,989,1725

cp,309,uy,998,1726
cp,310,uy,1001,1727
cp,311,uy,1004,1728

cp,312,uy,1013,1729
cp,313,uy,1016,1730
cp,314,uy,1019,1731
!****************************

!Now simulating carck by coupling in x and y direction
!First left joint
!****************************
cp,315,ux,451,461
!cp,316,ux,452,462
cp,317,ux,454,463

cp,318,uy,451,461
cp,319,uy,452,462
cp,320,uy,454,463

!****************************
cp,321,ux,514,1029
cp,322,ux,507,1030
cp,323,ux,497,1032

cp,324,uy,514,1029
!cp,325,uy,507,1030
cp,326,uy,497,1032
!****************************

!Next right joint
!****************************
cp,327,ux,965,975
!cp,328,ux,966,976
cp,329,ux,968,977

cp,330,uy,965,975
cp,331,uy,966,976
cp,332,uy,968,977
!****************************
!cp,333,ux,975,1713
cp,334,ux,982,1711
cp,335,ux,992,1710

!cp,336,uy,975,1713
!cp,337,uy,982,1711
cp,338,uy,992,1710
!****************************
!***** END OF COUPLING *****
```

```
!***** APPLYING BOUNDARY CONDITION *****

!First Left Column
!****************************
d,3,all,
d,6,all,
d,7,all,
d,8,all,
d,9,all,
d,10,all,
!****************************


!Now Right Column
!****************************
d,517,all,
d,520,all,
d,521,all,
d,522,all,
d,523,all,
d,524,all,
!****************************

cpdele,36

cp,36,ux,452,453,462


cpdele,205

cp,205,uy,507,1030,1031

cpdele,126

cp,126,ux,966,967,976

cpdele,327

cp,327,ux,965,975,1713


cpdele,330

cp,330,uy,965,975,1713

cpdele,296

cp,296,uy,982,1711,1712

!***** END OF APPLICATION OF BOUNDARY CONDITION *****

f,508,fx,5000

!***** HOPEFULLY THE END OF MODELLING *****

/solu
solve
```

```
/post1
nsel,s,node,,1006,1006
/output,results.txt,,,append
prnsol,u,x
/output
finish
/clear,nostart
```

# ANSYS Script File for Validation Model

```
parres,,param.txt

/prep7


!Drawing the nodes of left column at level 1

!**************************
n,1,0
n,2,1.5
n,3,1.5
n,4,4.75
n,5,8
n,6,8
n,7,11.25
n,8,14.5
n,9,14.5
n,10,16

n,11,0,2
n,12,1.5,2
n,13,8,2
n,14,14.5,2
n,15,16,2

ngen,30,15,1,15,,0,4
ngen,2,450,1,10,,0,120
!***************************

!Drawing nodes of right coulmn at level 1

!**************************
ngen,2,460,1,460,,240,0
!**************************

!Drawing nodes of left coulmn at level 2

!**************************
ngen,2,920,1,460,,0,140
!**************************

!Drawing nodes of right coulmn at level 2

!**************************
ngen,2,1380,1,460,,240,140
!**************************

!Summary of nodes of columns
!*************************
!Left Column         Level 1     Nodes 1      to     460
!Right Column        Level 1     Nodes 461    to     920
!Left Column         Level 2     Nodes 921    to     1380
!Right Column        Level 2     Nodes 1381   to     1840
!*************************


!Drawing nodes for beam at level 1

!***************************
```

```
n,1841,16,140
n,1842,16,137.5
n,1843,16,137.5
n,1844,16,133.75
n,1845,16,130
n,1846,16,130
n,1847,16,126.25
n,1848,16,122.5
n,1849,16,122.5
n,1850,16,120

n,1851,19.5,140
n,1852,19.5,137.5
n,1853,19.5,130
n,1854,19.5,122.5
n,1855,19.5,120

ngen,32,15,1841,1855,,7,0
ngen,2,480,1841,1850,,224,0
!****************************

!Drawing nodes of beam at level 2

!**************************
ngen,2,490,1841,2330,,0,140
!**************************

!Summary of nodes of beams
!**************************
!Lower beam Level 1      Nodes 1841  to    2330
!Upper beam Level 2      Nodes 2331  to    2820
!**************************


!Drawing nodes for left joint at level 1

!**************************
n,2821,0,120
n,2822,1.5,120
n,2823,4.75,120
n,2824,8,120
n,2825,11.25,120
n,2826,14.5,120
n,2827,16,120

n,2828,0,122.5
n,2829,1.5,122.5
n,2830,1.5,122.5
n,2831,1.5,122.5
n,2832,4.75,122.5
n,2833,8,122.5
n,2834,8,122.5
n,2835,8,122.5
n,2836,11.25,122.5
n,2837,14.5,122.5
n,2838,14.5,122.5
n,2839,14.5,122.5
n,2840,16,122.5

n,2841,0,126.25
n,2842,1.5,126.25
```

```
n,2843,8,126.25
n,2844,14.5,126.25
n,2845,16,126.25

n,2846,0,130
n,2847,1.5,130
n,2848,1.5,130
n,2849,4.75,130
n,2850,8,130
n,2851,8,130
n,2852,11.25,130
n,2853,14.5,130
n,2854,14.5,130
n,2855,16,130

n,2856,0,133.75
n,2857,1.5,133.75
n,2858,8,133.75
n,2859,14.5,133.75
n,2860,16,133.75

n,2861,0,137.5
n,2862,1.5,137.5
n,2863,1.5,137.5
n,2864,1.5,137.5
n,2865,4.75,137.5
n,2866,8,137.5
n,2867,8,137.5
n,2868,8,137.5
n,2869,11.25,137.5
n,2870,14.5,137.5
n,2871,14.5,137.5
n,2872,14.5,137.5
n,2873,16,137.5

n,2874,0,140
n,2875,1.5,140
n,2876,4.75,140
n,2877,8,140
n,2878,11.25,140
n,2879,14.5,140
n,2880,16,140
!**************************


!Drawing nodes of right joint at level 1

!**************************
ngen,2,60,2821,2880,,240,0
!**************************

!Drawing nodes of left joint at level 2

!**************************
ngen,2,120,2821,2880,,0,140
!**************************

!Drawing nodes of right joint at level 2

!**************************
ngen,2,180,2821,2880,,240,140
```

```
!**************************

!Summary of nodes of joints
!**************************
!Left joint Level 1     Nodes 2821  to    2880
!Right joint       Level 1     Nodes 2881  to    2940
!Left jont  Level 2     Nodes 2941  to    3000
!Right joint       Level 2     Nodes 3001  to    3060
!**************************

!!!!!! END OF NODE GENERATION !!!!!!

!Units definition

!****************************
/units,bin
!****************************

!!!!!! START OF ELEMENT, MATERIAL REALCONSTANT DEFINITION !!!!!!

!Defining Concrete Property

!****************************
mp,ex,concrete,3e6
mp,nuxy,concrete,.17
!****************************

!Defining Steel Property

!****************************
mp,ex,steel,30e6
mp,nuxy,steel,.3
!****************************

!Definition of element

!****************************
et,pl82c,plane82
keyopt,1,3,3
keyopt,1,5,0
keyopt,1,6,0
et,pl42c,plane42,,,3,,2,3
et,reinfc,link1
et,pl82b,plane82,,,3,,2,3
et,pl42b,plane42,,,3,,2,3
et,reinfb,link1
et,beamsprg,combin14,0,1                    !spring with UX dof, used in beam
et,colsprg,combin14,0,2                        !spring with UY dof, used in
column
!****************************
```

```
!Definition of Real Constants

!****************************
!Column element thickness plane82
!width of column
r,1,12

!Column element thickness plane42
!width of column
r,2,12

!Column reinforcement area divided by 3 link
!width of reinf. in column
r,3,COLRAREA

!Beam element thickness plane82
!width of beam
r,4,12

!Beam element thickness plane42
!width of column
r,5,12

!Beam element reinforcement link (actual area of either the top or
bottom reinf.)
!width of reinf
r,6,BMRAREA

!K value for spring used in beam
!value of k of the spring
r,7,2526.888*BRDIA*(BEAMSPAN/45)*NBBEAM*FT

!K value for spring used in column
!value of k of the spring
r,8,2526.888*CRDIA*(COLHT/30)*NBCOL*FT
!****************************


!!!!!! END OF ELEMENT, MATERIAL REALCONSTANT DEFINITION !!!!!!


!!!!!! START OF ELEMENT GENERAITION !!!!!!

!Left Column

!Generating Concrete Elements

!****************************
type,pl82c
mat,concrete
real,pl82c
!****************************

!First 8 noded elements
!****************************
e,2,5,20,17,4,13,19,12
e,5,8,23,20,7,14,22,13
!****************************
```

```
!****************************
type,pl42c
mat,concrete
real,pl42c
!****************************

!Now 4 noded elements
!****************************
e,1,2,12,11
e,8,10,15,14
e,11,12,17,16
e,14,15,25,23
!****************************


!****************************
type,reinfc
mat,steel
real,reinfc
!****************************

!Now generating the reinforcement
!****************************
e,3,18
e,6,21
e,9,24
!****************************


!****************************
type,colsprg
mat,steel
real,colsprg
!****************************

!Now generating the spring
!****************************
e,2,3
e,5,6
e,8,9
!****************************


!Copying the elements generated so far
!****************************
egen,30,15,1,12
!****************************


!****************************
type,colsprg
mat,steel
real,colsprg
!****************************

!The last layer of spring in the left column
!****************************
e,452,453
e,455,456
e,458,459
!****************************
```

```
!Drawing elements for right column at level 1
!****************************
egen,2,460,1,363
!****************************

!Drawing elements for left column at level 2
!****************************
egen,2,920,1,363
!****************************

!Drawing elements for right column at level 2
!****************************
egen,2,1380,1,363
!****************************


!Summary of elements of columns
!**************************
!Left Column        Level 1     Elements    1       to      363
!Right Column       Level 1     Elements    364     to      726
!Left Column        Level 2     Elements    727     to      1089
!Right Column       Level 2     Elements    1090    to      1452
!**************************


!Drawing elements for beam at level 1
!****************************

!****************************
type,pl82b
mat,concrete
real,pl82b
!****************************

!First 8 noded elements
!****************************
e,1848,1863,1860,1845,1854,1862,1853,1847
e,1845,1860,1857,1842,1853,1859,1852,1844
!****************************


!****************************
type,pl42b
mat,concrete
real,pl42b
!****************************

!Now 4noded elements
!****************************
e,1850,1855,1854,1848
e,1855,1865,1863,1854

e,1842,1852,1851,1841
e,1852,1857,1856,1851
!****************************


!****************************
type,reinfb
mat,steel
```

```
real,reinfb
!****************************

!Reinforcements
!****************************
e,1849,1864
!e,1034,1049
e,1843,1858
!****************************


!****************************
type,beamsprg
mat,steel
real,beamsprg
!****************************

!Springs
!****************************
e,1848,1849
!e,1033,1034
e,1842,1843
!****************************


!Copying the elements generated so far
!*************************************
egen,32,15,1453,1462
!*************************************


!Generating the last layer of spring in the beam

!****************************
e,2322,2323
!e,1708,1709
e,2328,2329
!****************************



!Drawing elements of beam at level 2

!**************************
egen,2,490,1453,1774
!**************************


!Summary of elements of beams
!**************************
!Lower beam Level 1     Elements    1453  to    1774
!Upper beam Level 2     Elements    1775  to    2096
!**************************


!Drawing elements for right joint at level 1
!****************************

!****************************
type,pl42c
```

```
mat,concrete
real,pl42c
!****************************

!4noded elements
!****************************
e,2821,2822,2829,2828
e,2822,2823,2832,2829
e,2823,2824,2833,2832
e,2824,2825,2836,2833
e,2825,2826,2837,2836
e,2826,2827,2840,2837

e,2828,2829,2842,2841
e,2841,2842,2847,2846
e,2846,2847,2857,2856
e,2856,2857,2862,2861

e,2837,2840,2845,2844
e,2844,2845,2855,2853
e,2853,2855,2860,2859
e,2859,2860,2873,2870

e,2861,2862,2875,2874
e,2862,2865,2876,2875
e,2865,2866,2877,2876
e,2866,2869,2878,2877
e,2869,2870,2879,2878
e,2870,2873,2880,2879

!****************************


!****************************
type,pl82c
mat,concrete
real,pl82c
!****************************

!8noded elements
!****************************
e,2829,2833,2850,2847,2832,2843,2849,2842
e,2833,2837,2853,2850,2836,2844,2852,2843
e,2847,2850,2866,2862,2849,2858,2865,2857
e,2850,2853,2870,2866,2852,2859,2869,2858
!****************************

!****************************
type,reinfc
mat,steel
real,reinfc
!****************************

!Link elements
!****************************
e,2830,2848
e,2848,2863
!e,28,28

e,2834,2851
e,2851,2867
```

```
e,2838,2854
e,2854,2871
!*****************************

!*****************************
type,colsprg
mat,steel
real,colsprg
!*****************************

!Spring elements
!*****************************
e,2829,2830
e,2847,2848
e,2862,2863

e,2833,2834
e,2850,2851
e,2866,2867

e,2837,2838
e,2853,2854
e,2870,2871
!*****************************

!*****************************
type,reinfb
mat,steel
real,reinfb
!*****************************

!Reinforcements
!*****************************
e,2831,2835
e,2835,2839

e,2864,2868
e,2868,2872
!*****************************

!*****************************
type,beamsprg
mat,steel
real,beamsprg
!*****************************

!Springs
!*****************************
e,2829,2831
e,2833,2835
e,2837,2839

e,2862,2864
e,2866,2868
e,2870,2872
!*****************************

!Drawing elements of right joint at level 1
```

```
!**************************
egen,2,60,2097,2145
!**************************

!Drawing nodes of left joint at level 2

!**************************
egen,2,120,2097,2145
!**************************

!Drawing nodes of right joint at level 2

!**************************
egen,2,180,2097,2145
!**************************



!Summary of elements of joints
!**************************
!Left joint Level 1      Elements    2097  to    2145
!Right joint      Level 1     Elements    2146  to    2194
!Left jont  Level 2     Elements    2195  to    2243
!Right joint      Level 2     Elements    2244  to    2292
!**************************


!Coonecting joints to column
!**************************

!L1 left joint + L1 left column
!**************************

!****************************
type,reinfc
mat,steel
real,reinfc
!****************************

!Link elements
!****************************
e,453,2830
e,456,2834
e,459,2838
!****************************

!L1 left joint + L2 left column
!**************************
!Link elements
!****************************
e,923,2863
e,926,2867
e,929,2871
!****************************

!L2 left joint + L2 left column
!**************************
!Link elements
!****************************
e,1373,2950
```

```
e,1376,2954
e,1379,2958
!*****************************

!L1 Right joint + L1 right column
!**************************
!Link elements
!*****************************
e,913,2890
e,916,2894
e,919,2898
!*****************************

!L1 right joint + L2 right column
!**************************
!Link elements
!*****************************
e,1383,2923
e,1386,2927
e,1389,2931
!*****************************

!L2 right joint + L2 right column
!**************************
!Link elements
!*****************************
e,1833,3010
e,1836,3014
e,1839,3018
!*****************************


!*****************************
type,reinfb
mat,steel
real,reinfb
!*****************************

!L1 left joint + L1 beam
!***************************
e,1843,2872
e,1849,2839
!***************************

!L1 right joint + L1 beam
!***************************
e,2323,2924
e,2329,2891
!***************************

!L2 left joint + L2 beam
!***************************
e,2333,2992
e,2339,2959
!***************************

!L2 right joint + L2 beam
!***************************
e,2813,3044
e,2819,3011
!***************************
```

```
!!!!!!! END OF ELEMENT GENERATION !!!!!!!!


!***** APPLYING BOUNDARY CONDITION *****

!First Left Column
!****************************
!d,3,all,
!d,6,all,
d,7,all,
d,8,all,
d,9,all,
d,10,all,
!****************************


!Now Right Column
!****************************
!d,463,all,
!d,466,all,
d,467,all,
d,468,all,
d,469,all,
d,470,all,
!****************************


!***** APPLYING COUPLING OF CO-INCIDENT NODES *****

!Coupling nodes of L1 left column in x direction
!****************************
nsel,s,node,,1,460
cpintf,ux
!****************************

!Coupling nodes of L1 right column in x direction
!****************************
nsel,s,node,,461,920
cpintf,ux
!****************************

!Coupling nodes of L2 left column in x direction
!****************************
nsel,s,node,,921,1380
cpintf,ux
!****************************

!Coupling nodes of L2 right column in x direction
!****************************
nsel,s,node,,1381,1840
cpintf,ux
!****************************

!Coupling nodes of L1 beam in y direction
!****************************
nsel,s,node,,1841,2330
cpintf,uy
!****************************
```

```
!Coupling nodes of L2 beam in y direction
!****************************
nsel,s,node,,2331,2820
cpintf,uy
!****************************

nsel,s,node,,2821,3060
cpintf,all

nsel,s,node,,1,3060

!***** Simulating Cracks *******
!L1-left joint + left column
cp,650,ux,451,2821
cp,651,uy,451,2821
cp,652,ux,452,2822
cp,653,uy,452,2822
cp,654,ux,454,2823
cp,655,uy,454,2823

!L1-left joint + left column
cp,656,ux,927,2878
cp,657,uy,927,2878
cp,658,ux,928,2879
cp,659,uy,928,2879
cp,660,ux,930,2880
cp,661,uy,930,2880

cp,662,ux,1841,2880
cp,663,uy,1841,2880
cp,664,ux,1842,2873
cp,665,uy,1842,2873
cp,666,ux,1844,2860
cp,667,uy,1844,2860

cp,668,ux,911,2881
cp,669,uy,911,2881
cp,670,ux,912,2882
cp,671,uy,912,2882
cp,672,ux,914,2883
cp,673,uy,914,2883

cp,674,ux,1387,2938
cp,675,uy,1387,2938
cp,676,ux,1388,2939
cp,677,uy,1388,2939
cp,678,ux,1390,2940
cp,679,uy,1390,2940

cp,680,ux,2327,2901
cp,681,uy,2327,2901
cp,682,ux,2328,2888
cp,683,uy,2328,2888
cp,684,ux,2330,2881
cp,685,uy,2330,2881

cp,686,ux,1371,2941
cp,687,uy,1371,2941
cp,688,ux,1372,2942
cp,689,uy,1372,2942
cp,690,ux,1374,2943
```

```
cp,691,uy,1374,2943

cp,692,ux,2331,3000
cp,693,uy,2331,3000
cp,694,ux,2332,2993
cp,695,uy,2332,2993
cp,696,ux,2334,2980
cp,697,uy,2334,2980

cp,698,ux,1831,3001
cp,699,uy,1831,3001
cp,700,ux,1832,3002
cp,701,uy,1832,3002
cp,702,ux,1834,3003
cp,703,uy,1834,3003

cp,704,ux,2817,3021
cp,705,uy,2817,3021
cp,706,ux,2818,3008
cp,707,uy,2818,3008
cp,708,ux,2820,3001
cp,709,uy,2820,3001

!cp,67,all,

cpdele,374,570,3

f,2966,fx,5000

cpdele,652
cpdele,27
cpdele,658
cpdele,253
cpdele,662
cpdele,660
cpdele,663
cpdele,661
cpdele,665
cpdele,373
cpdele,670
cpdele,120
cpdele,676
cpdele,346
cpdele,683
cpdele,471
cpdele,684
cpdele,668
cpdele,685
cpdele,669
cpdele,688
cpdele,213
cpdele,695
cpdele,472
cpdele,700
cpdele,306
cpdele,707
cpdele,570
cpdele,708
cpdele,698
cpdele,709
cpdele,699
```

```
cp,next,ux,452,453,2822
cp,next,ux,928,929,2879
cp,next,ux,930,1841,2880
cp,next,uy,930,1841,2880
cp,next,uy,1842,1843,2873
cp,next,ux,912,913,2882
cp,next,ux,1388,1389,2939
cp,next,uy,2328,2329,2888
cp,next,ux,911,2330,2881
cp,next,uy,911,2330,2881
cp,next,ux,1372,13732942
cp,next,uy,2332,2333,2993
cp,next,ux,1832,1833,3002
cp,next,uy,2818,2819,3008
cp,next,ux,1831,2820,3001
cp,next,uy,1831,2820,3001
```

# REFERENCES


1. **ACI Committee 435** – Control of Deflection in Concrete Structures, *ACI Committee Report ACI 435R-95*, p. 435R-1, 1995.
2. **ACI-ASCE Committee 352** - Recommendations for Design of Beam-Column Joints in Monolithic Reinforced Concrete Structures, *ACI Committee Report ACI352R-91*, p. 352R-1, 1991
3. **Amanat K.M., Ahmed B., Hoq E., Enam M.B.** - Computational Investigation of the Semi-Rigid Properties of a RC Beam-Column Joint, *Proceedings of the Civil and Environmental Engineering Conferences-New Frontiers and Challenges*, AIT, Bangkok, Thailand, November, 1999.
4. **Amanat K. M., Enam M. B.** - Study of the Semirigid Properties of Reinforced Concrete Beam-Column Joints, *Journal of Civil Engineering, The Institution of Engineers, Bangladesh*, Vol. CE 27, No. 1, 1999
5. **Ahsan R.** - *Semi-rigid Joint Action in Sway Steel Frames*, M.Sc. thesis, Dept. of Civil Engineering, BUET, 1997.
6. **Amanat K.M.** - A Design Rationale for Free Standing Stair Slabs Based on Finite Element Analysis, *M.Sc. thesis*, Dept. of Civil Engineering, BUET, 1993.
7. **Alameddine F.** and **Ehsani M.R.** - High-Strength RC Connections Subjected to Inelastic Cyclic Loading, *American Soc. of Civ. Eng. (ASCE), Journal of Structural Engineering*, Vol. 117, No. 3, p. 829-850,1991.
8. **Amanat K.M.** - *Gradient Plasticity Modeling for the Fracture and Strain Localization of Early Age Concrete*, Ph.D. thesis, Dept. of Civil Engineering, Nagoya University, Japan, 1997.
9. **Sheikh T. M., Deierlein G. G., Yura J. A.** and **Jirsa J. O.** - Beam-Column Moment Connections for Composite Frames, *American Soc. of Civ. Eng. (ASCE), Journal of Structural Engineering*, Vol. 115, No. 11, November, 1989.

10. **Mehrabi A. B., Shing P. B., Schuller, M. P.** and **Noland J. L.** - Experimental Evaluation of Masonry-Infilled RC Frames, *American Soc. of Civ. Eng. (ASCE), Journal of Structural Engineering,* Vol. 122, No. 3, March, 1996.
11. **Baker J.F.** - A Note on the Effective Length of a Pillar Forming Part Of a Continuous Member in a Building Frame, *2nd Report,* Steel Structure Research Committee, Dept. of Scientific and Industrial Research, p. 13-44, HMSO, London, 1934.
12. **Batho C.** and **Rowan H.C.** - Investigations on Beam and Stanchion Connections, *2nd Report,* Steel Structure Research Committee, Dept. of Scientific and Industrial Research, p. 92, HMSO, London, 1934.
13. **Chen W.F.** and **Kishi** N. - Semi-rigid steel Beam-to-column Connections: Data Base and Modeling, *American Soc. of Civ. Eng. (ASCE), Journal of Structural Engineering,* Vol. 115, No. 2, p. 105-119, 1986.
14. **Goverdhan** A.V. - *A Collection of Experimental Moment-Rotation Curves and Evaluation of Prediction Equations for Semi-rigid Connections,* Master's Thesis, Vanderbilt University, Nashville TN, p. 490.
15. **Jones S.W., Kirby P.A.** and **Nethercot D.A.** - Modeling of Semi-rigid Connection Behaviour and its Influence on Steel Column Behaviour, In *Joints in Structural Steelwork,* p. 5.7-5.87, Pentech Press, London, 1981.
16. **Jones S.W., Kirby P.A.** and **Nethercot D.A.** - Columns with Semi-rigid Joints, *American Soc. of Civ. Eng. (ASCE), Journal of Structural Engineering,* Vol. 108, p. 361-372, 1982.
17. **Qi X.** and **Pantazopoulou** S. **J.** - Response of RC Frame under Lateral Loads, *American Soc. of Civ. Eng. (ASCE), Journal of Structural Engineering,* Vol. 117, No. 4, p. 1167-1188, 1991.
18. **Rathbun J.C.** - Elastic Properties of Riveted Connections, *American Soc. of Civ. Eng. (ASCE), Journal of Transportation Engineering,* Vol. 101, p. 524-563,1936.