EFFECT OF IN-PLANE FORCES IN BEAM-COLUMN JUNCTION OF RC SUBSTITUTE FRAME IN THE LINEAR REGIME

Amiya Kr. Samanta and S. Tripathi
Department of Civil Engineering, National Institute of Technology Durgapur, India
E-Mail: aksnitd@gmail.com

ABSTRACT

The practicing professionals mostly prefer to employ approximate models for the purpose of design of a structural domain. Model based on such method ignores a designer to evaluate the deformation pattern of the structure as a whole, as the actual 3D problems are over-simplified into 2D problems using line elements based on certain assumptions. Thus a designer handles a problem keeping him in darkness in these areas, rather adopts a conservative approach in view of quick and easy solution for the purpose at hand. Moreover reinforcement requirement remains on much higher side, thereby increasing hazards in detailing as well as construction site along with overall cost of the structure. Also in earlier days, sophisticated tools like various FEM packages were not also available commercially and easily in the market. Hence frames/structures were analyzed as grid structure/wire mesh. Although a wire mess gives some idea about the deformation pattern, but it's unable to handle the complicated situations. It has been shown with the aid of present study of RC substitute frame that the values of bending stresses obtained thus are exceptionally on conservative side compared to the analysis output using FEM package, e.g. Abaqus. Here the behaviour of a substitute frame has been presented, following three different methods. Various results obtained have been compared to assess their potentiality and suitability in understanding true behaviour of such a system.

Keywords: substitute frame, beam-column joint, bending, membrane effect, STAADPro, abaqus.

INTRODUCTION

It has been observed that if an under reinforced beam which is restrained develops some cracks (at the point of maximum sagging and at the supports) due to excessive loading, axial forces are developed. In a frame where the beam is restrained by columns, the axial forces developed due to cracking of beam come on to the column, which some times results in brittle failure for the column. Brittle failure is extremely dangerous and is never preferred in designing of concrete structures as the failure occurs all of a sudden and there is no time for saving lives and things coming beneath the structure. Brittle failure generally occurs in columns due to high axial loads coming on to it. In comparison to this, beams generally go through ductile failure. All the designers have always aimed that if the failure occurs then it should be ductile failure only.

It has been observed that generally a frame fails due to two reasons (1) Ductile failure of the beams or (2) Brittle failure of columns. The failure solely depends on the end restraint provided to the beam by the column. In case of a beam connected by columns at the ends, the lateral reactions will try to oppose the bending of the beam and so the deflection will be reduced to some extent. These lateral forces that try to pull back the beam to remain straight, are called membrane /in-plane forces. Membrane forces are developed in beams depending on the end restrained conditions. This is shown in the diagram below. These forces result in increase of strength of the beams as these forces try to minimize the deflection. But due to these forces, moment is developed on the column that is in connection to the beam. These moments are not small that they can be neglected.

In fact this extra moment could result in increasing the total moment coming on the column by about 7 times the total moment coming on the column as calculated by ignoring the membrane forces. Sometimes this extra moment generated due to the membrane forces, results in brittle failure of the column. This in turn results in failure of the members, which are supported above the column, which may lead to the ultimate collapse of the building. The consequences are even more if this brittle failure occurs at the column of end span or at the periphery of the building. It will straightway lead to sudden collapse of the building.

Over the past 40 years the phenomenon of compressive membrane action has been the subject of a number of research studies with a view to producing more economical design procedures, particularly for slabs. This phenomenon was evidently recognized by the pioneers of slab construction. In later years, experimental investigations have been carried out on a complete building, on series of slabs in laboratory research programmes and on un-reinforced brick beams constrained between rigid supports. They all showed that the ultimate loads for flexural failure can exceed those predicted by normal design methods. In a fundamental study on the elastic and plastic method of slab design, a modified yield criterion was predicted for laterally restrained slabs, based on the interaction of membrane force and bending moment derived from rigid plastic theory. This theory marked the real beginning of the research for a rational method of incorporating compressive membrane effects into the analysis of reinforced concrete slabs.

The membrane action has been neglected till now while designing the frames. And so we can very confidently say that the present designs are not up to the

VOL. 4, NO. 10, DECEMBER 2009 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

mark and are not safe to the highest level. There is an immediate need to modify the design procedures, especially for frames so that our designs can be more accurate in catering the loads, more economical and more durable. Before doing that we need to study properly the frame structures comprising of beams and columns and their junction so that we can see what extra forces are being developed on different members, where they are being developed and how they are being developed.

So here an investigation work has been taken up to study this effect in case a frame and design the members accordingly for comparison. For studying the frame a substitute frame subjected to two transverse loads along with its self-weight. The same has been analyzed following three methods to look at the gap the designers usually overlook.

Method-I: Analytical method (moment distribution)

Method-II: Software like STAAD PRO (matrix method of analysis)

Method-III: Software like Abaqus (3D FEM analysis)
Since nowadays, high speed computing facilities and high quality commercial softwares are also available, the investigators as well as professionals may go for solution of complicated 3D problems of varieties in general. 3D modeling may be a potential approach in order to achieve more realistic solution in general. In the present case, the 3D linear hexahedral lower order element C3D8I of Abaqus software, which uses incompatible modes, has been used to model the frame following Method - III. This paper simulates the elastic response of the substitute frame considering (1) concrete as a solid isotropic homogeneous medium, which uses linear elasticity based constitutive model, (2) lower order solid elements to represent concrete medium, which reduces time and associated cost in terms of easy and simple mesh generation together with data interpretation. Also a good effort has been attributed to develop a comparative study with the standard analytical methods e.g. moment distribution method and matrix method of analysis.

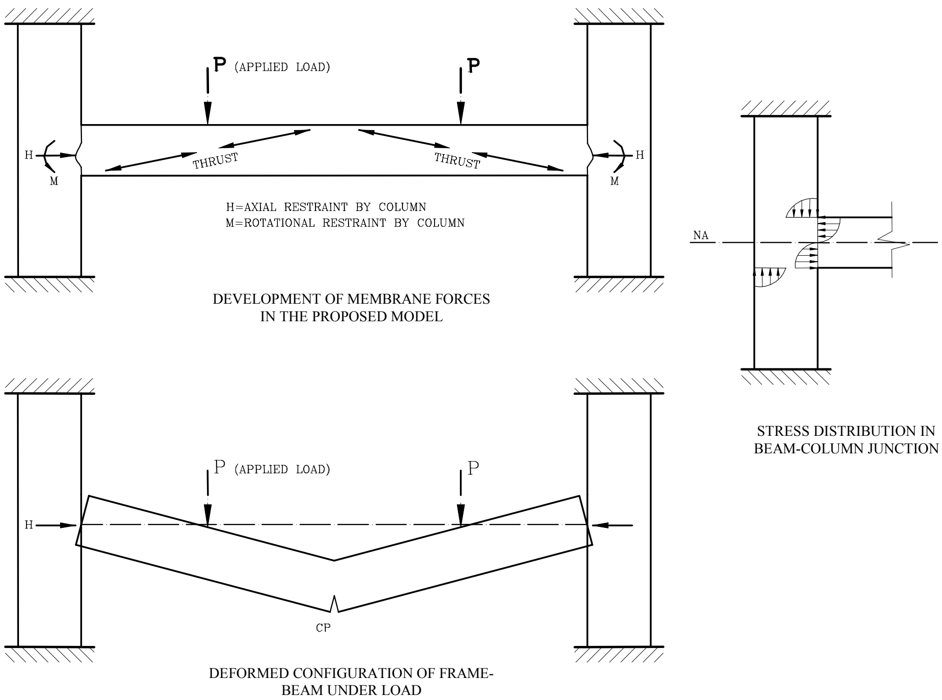

Figure-1. Behaviour of substitute frame.

PROBLEM LAYOUT AND OBJECTIVE

A single bay substitute frame as shown in Figure-2 with column section 125 (b) x 250 (d) and beam section 125 (b) x 200 (d) has been investigated. The beam is 2.0m long (clear span) and is subjected to two point loads (4.0 MT each) at quarter span of the beam only apart from its self-weight. The concrete has the characteristic strength of 25Mpa, elastic modulus (E_c) = 25000 MPa, Poisson's ratio (μ). = 0.17. The reinforcement bar has the elastic modulus Es = 200000 MPa with effective cover (d') on both sides equal to 50mm. The investigation is concerned with the study of beam and column junction. Here three different ways of analysis are adopted to calculate the moment at different sections. The problem consists of three parts; viz. the above frame has been analyzed considering the line elements along the centroid of each section for its components using moment distribution method.

Also there is a need to use some software to reach a more precise result and check the results as obtained by the moment distribution method. This method of analysis considers the moment distribution factor assuming the beam as a flexural member and column as axially loaded

VOL. 4, NO. 10, DECEMBER 2009 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

flexural member, but it does take into account of the axial force the beam may carry, i.e. it neglects the beam-column effect. So next level refinement has been done by using other matrix methods of analysis (STAADPro) as well to cater the same effect. Again the span of beam has been modified by considering the centerline length for the above two methods and hence we are unable to pick up the actual distribution of moment /shear at the junction.

Hence, further to make a deep study into the beam-column junction and the way the junctions caters the moment coming on to it, we will go for another way of analysis using another software-Abaqus. This software uses finite element method to analyze the structure. So we can obtain the moment coming on even small elements of the beam-column system. Moreover this software uses the 3D model of the structure, which helps in getting more precise result. By this method we will get the exact values of moments at the junction and other sections. As this method is using the 3D model so the value of the moments at the junctions will be equivalent to the actual moments coming on the beam-column junction.

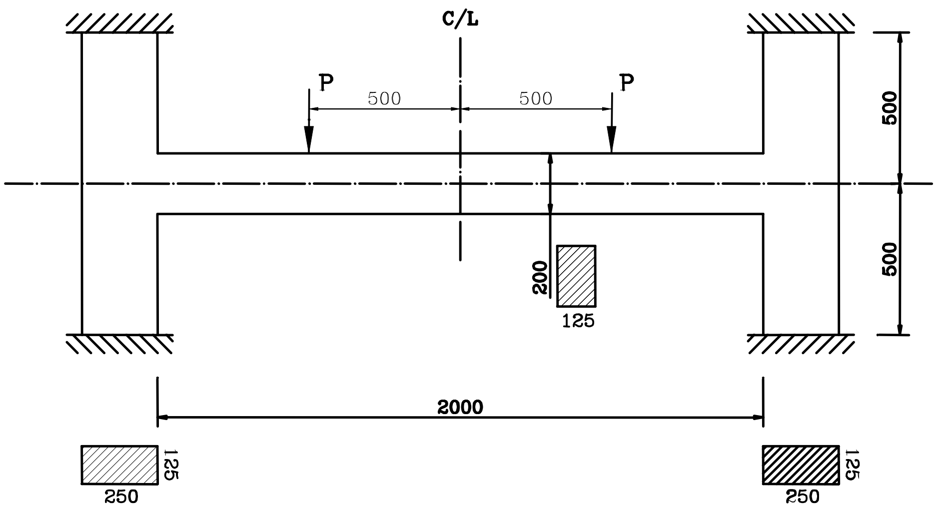

Figure-2. Single by substitute frame.

The basic purpose of choosing this problem of failure of beam, column or the beam column junction, is to study the failure pattern and know what are the factors, which lead to these kinds of failure. Also now when we know about the membrane forces, which develop in a frame, we should study how it affects the beams and columns, which form a frame. Generally when we find out the values of moment being developed over a member of a frame comprising of beam and column junction, we use the centerline diagram. We also find out the moment coming on to the beam and column junction by using the same centerline diagram. The value we get at the junction is then used for the design purpose of beam and column. But in reality the moment coming on the junction of beam and column is somewhat different as by using centerline diagram we use the centre line of beams and columns. So the junctions of those lines are not the junction of beam and column in reality. And so the moments we are using for the design of beams and columns are the wrong values. Thus there is a need to study what moment is actually coming on the junction of beam and column and what is the difference of the actual moment coming and the moment we are calculating.

Earlier studies have shown that the membrane forces developed strengthens the beams but also brings some extra moment on the columns. The strengthening of beams totally depends up on the end-restrained conditions of the beams. If the beams are restrained by slab panels or some stiff members, the increase in strength of the beams becomes very large. But if columns, which are considered to be a bit flexible, restrain the beams the increase in strength of the beams is very small. So more the stiff end-restraining member of beam will be, more will be the increase of strength of beam. And more the increase in strength of the beam will be more will be the moment coming on the column. Many past studies have shown that the failure of such kind of frame was mainly due to the ductile failure of the beam or the brittle failure of the either columns. After finding out the moments at different sections, diagram derived from Abaqus indicates the moment values at different sections of the frame. Abaqus results give us a real picture of the bending pattern and the values of moments coming on different members. As takes into account the membrane action, it is able to provide the true nature of the deformation pattern as well.

RESULTS AND DISCUSSIONS

The underlying purpose of choosing this problem of frame is to study the effect and restraint provided by the beam-column junction in predicting the response general building frame structures. That is to say, to identify and evaluate the contribution of the membrane forces which develops in the beam-column junction a frame. As stated earlier, the above-mentioned frame is analyzed using three methods. For the first two methods i.e. the moment distribution method and using convention analysis software e.g. STAADPro, which uses only matrix method of analysis, the considered frame has been converted into a frame consisting of line elements along the centre line of each of the members as shown in Figure-3.

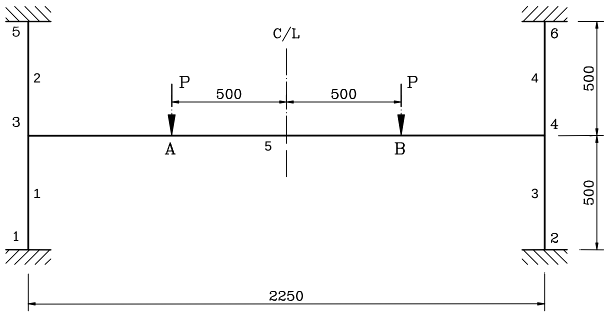

Figure-3. Frame for mthod-1 and 2.

The conventional /manual method of finding out the moments through moment distribution method has been followed, which is basically a force method of solution of structure. The relative stiffnesses of all the members are found out by dividing the moment of inertias of respective members with their lengths. The total stiffness of a joint is then calculated by adding all the relative stiffness of different members attached to that joint. The distribution factor is found out later on for each member by dividing the relative stiffness of that member by the total stiffness of the joint to which that member is attached. These distribution factors are used to derive the final moments for different members.

With the growing concern of the practicing engineers to achieve faster solution of analysis results to survive in the competitive market, there is a tendency to use commercial softwares e.g. Structural Analysis and Design i.e. STAAD, which follows stiffness method. It is one of the first software applications in the world made for the purpose of helping the structural engineers to automate their work, to eliminate the tedious and lengthy procedures of the manual methods. In this context, that's why the same software has been used to analyze this simple frame for the assessment of bending moment diagram. In STAAD Pro GUI environment, all the required inputs were provided regarding load, boundary conditions, material properties (modulus of elasticity, material type etc.). While working on STAADPro, some of the inputs were supplied regarding the properties of the material used. The material was defined as isotropic concrete, Modulus of elasticity (E) = 2.17185E+007 kN/m^2, Poisson ratio (μ) = 0.17, Density (ρ) = 23.5616 kN/m^3, nodes 1,2,5,6 were considered as fixed. One of the loads was applied at 1.625m from the left hand side beam column junction and the other at 0.625m, respectively as shown in Figure-3.

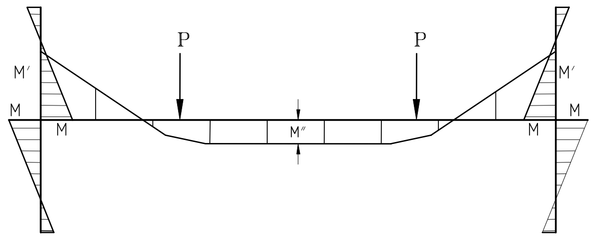

Figure-4. BMD for method-1 and 2.

The above substitute frame has also been analyzed using the FEM software Abaqus to compare the prediction of load-deformation response by previous two methods for the purpose. This commercial software is well established for the stress analysis in three-dimensional domain. The element mess was generated by aspect ratio close to unity and deviation factor equal to 0.1 and all input were in mm and N (SI). Such a procedure divides the width (250mm) of the column into 3 equal divisions, height (1000mm) of the column into 10 equal divisions, depth (200mm) of the beam into 2 equal divisions and length (1000mm) of the beam into 20 equal divisions. Thus it generates total 290 nodes and 100 elements. Figure-4 shows the mess layout along with node levels and Figure-5 shows the element incidences. It uses the lower order hexahedral and linear 3D stress elements (type C3D8I) using the incompatible mode. The column ends have been considered with fixed boundary condition. The frame has only been analyzed for linear elastic condition using concrete with material properties; modulus of elasticity E = 25000 MPa as per the IS code practice for plain /reinforced concrete and Poison's ratio μ = 0.17 (for M25 grade of concrete). The frame material has been considered as isotropic/homogeneous and the rebar contribution has not been included in the analysis.

VOL. 4, NO. 10, DECEMBER 2009 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

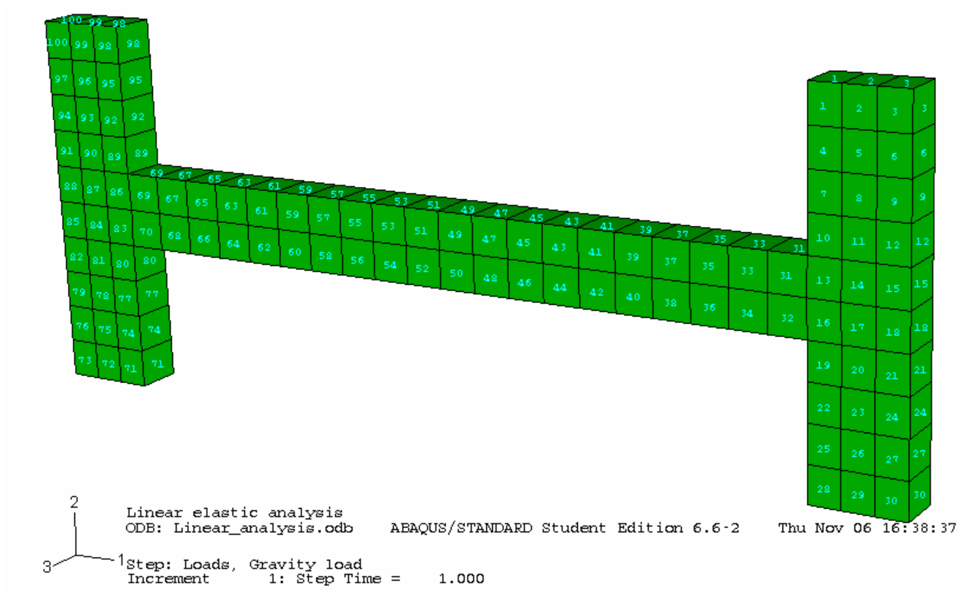

Figure-5. Mess layout of frame (Abaqus): element levels.

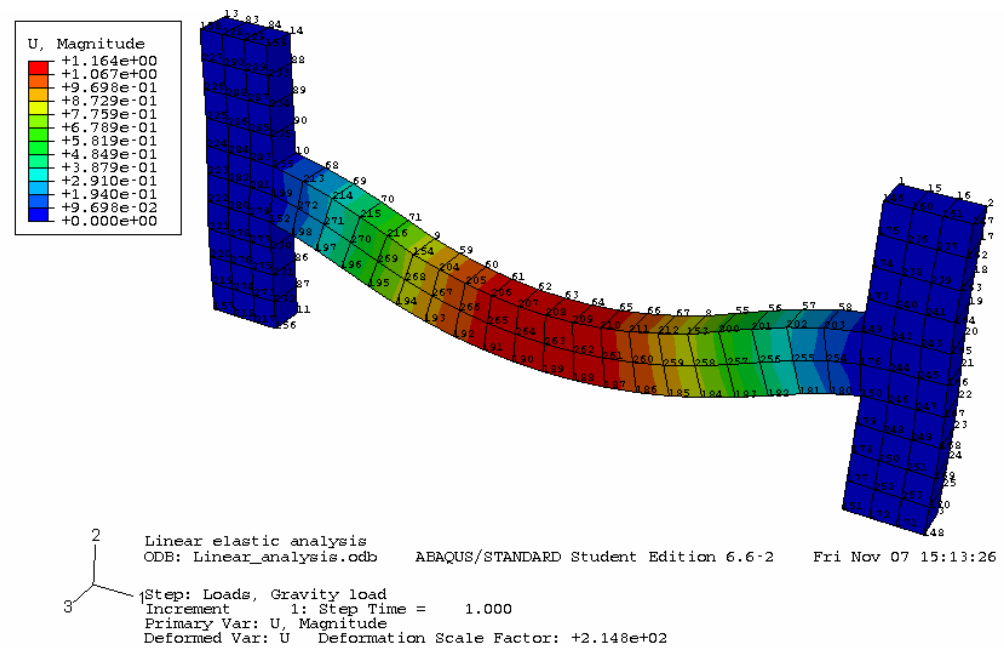

Figure-6. Deformed configuration of frame (Abaqus): deflection.

As far as loading is concerned, only self-weight /gravity load along with the point loads as shown in Figure-2, has been taken into consideration in the elastic regime using Abaqus. The deformed configuration with the deflection (vertical) spectrum and stress contours has been plotted in Figures 6 and 7, respectively. The maximum mid-span deflection from the Abaqus frame analysis has been noted as 1.164 mm. Also it clearly depicts that the beam is stressed most at and near the supports of the beam (12.67 MPa), although the maximum deflection is taking place at the center of the beam.

VOL. 4, NO. 10, DECEMBER 2009 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

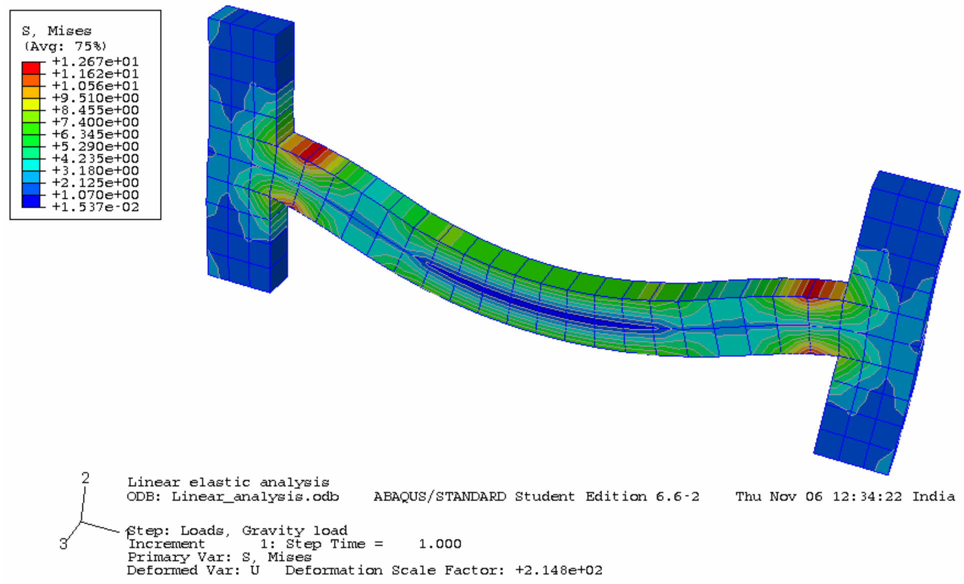

Figure-7. Deformed configuration of frame (Abaqus): stress.

Comparison of stress and deflection

The results obtained following three methods have been listed herewith the critical /design values so as to assess the potentiality, reliability of such methods in predicting behaviour of such structural system. Following Method-1, which is a conventional, analytical method and which doesn't consider the effect of interaction of the members in terms of axial forces, the critical values of bending moment at the beam-column junction (M') and at mid span (M") of the beam have been observed as 8.55kNm and 3.75kNm, respectively. The revised moment value at the column face i.e. with face correction (M'R) has been derived as 6.05kNm.

Following Method-2 i.e. STAADPro, which is mostly followed it has been observed that M = 8.60kNm, Mc = kNm and M'= kNm respectively. The moment value at the column face (M') has been derived as 6.05kN-m. As the effects of membrane forces are usually neglected in analytical methods and even the STAADPro fails to capture the same although very often preferred by the practicing engineers.

Table-1. Comparison of stress.

Method	At support				At mid span	
	M' (kNm)	Stress σ (MPa)	M'_R (kNm)	Stress σ'_R (MPa)	M" (kNm)	Stress σ" (MPa)
1.	8.55	10.26	6.05	7.26	3.75	4.5
2.	8.60	10.32	6.1	7.32	3.9	4.68
3.	--	--	--	8.49	--	6.37

It is obvious from Table-1 that the stress values for Method-1 and Method-2 are very close to each other, whereas for the Method-3 it grossly varies. The value as obtained from moment distribution method is very close to the value obtained from STAAD Pro. But on the other hand the value is far less as obtained from Abaqus, at the beam-column junction in particular. As Abaqus uses 3D stress analysis, it represents a true load-deformation behaviour close to the actual, of course neither of the above three methods uses the additional stiffness contribution from reinforcements. The rigidity of the columns arrests the rotation of beam at each support with the development of membrane forces at the junction. The stress at the junction becomes less than the same provided by Method-1 and 2, but higher than the corrected value derived at the column face.

Many a times, practicing engineers apply a correction to the support moment values for the design of beams at the face of the column. As this investigation suggests that actually the designers' underestimates the support moment values, whenever they apply such corrections. As the Abaqus analysis indicates, actual behaviour of the junction remains in between the analysis moment and corrected moment values. Also as a result, span moment values are not properly taken care of. In fact, a certain amount of redistribution of support moment derived from conventional analysis methods may lead to appropriate behaviour, which is shown by Abaqus. For the sake of comparison of deflection, Table-2 has been prepared, which shows the deflection of a number of points along the center line of the frame beam following conventional method of analysis (STAADPro), Abaqus and the proposed model.

Also Figure-8 shows the same comparison pictorially and it is observed that the model predictions following Method-2 (conventional STAADPro analysis) differs substantially from analysis output following Method-3 (Abaqus).

VOL. 4, NO. 10, DECEMBER 2009 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

Table-2. Comparison of deflection of frame beam.

Sl. No.	STAADPro		ABAQUS		
	l = 2250 mm		l = 2000 mm		
	length (mm)	deflection (mm)	Node no.	length (mm)	deflection (mm)
1	0	-0.015	199	125	-0.0628
2	112.5	-0.115	272	225	-0.152
3	225	-0.305	271	325	-0.2895
4	337.5	-0.553	270	425	-0.4592
5	450	-0.827	269	525	-0.6532
6	562.5	-1.097	268	625	-0.8226
7	675	-1.334	267	725	-0.9359
8	787.5	-1.52	266	825	-1.0492
9	900	-1.654	265	925	-1.0982
10	1012.5	-1.734	264	1025	-1.163
11	**1125**	**-1.7457**	263	**1125**	**-1.164**
12	1237.5	-1.734	262	1225	-1.163
13	1350	-1.654	261	1325	-1.0982
14	1462.5	-1.52	260	1425	-1.0492
15	1575	-1.334	259	1525	-0.9359
16	1687.5	-1.097	258	1625	-0.8226
17	1800	-0.827	257	1725	-0.6532
18	1912.5	-0.553	256	1825	-0.4592
19	2025	-0.305	255	1925	-0.2895
20	2137.5	-0.115	254	2025	-0.152
21	2250	-0.015	176	2125	-0.0628

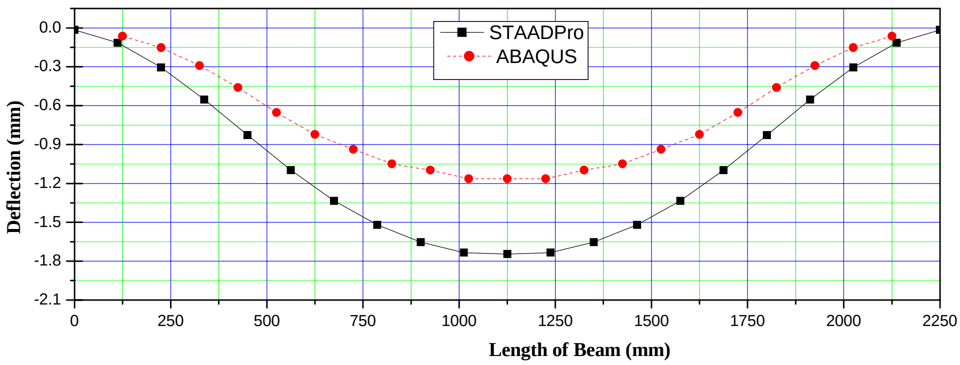

Figure-8. Deflection plot of frame beam.

The results of the analysis of the frame taken into consideration by all the 3 methods has made it very clear that membrane forces affects the strength of beam and column of a frame. It strengthens the beam and results into action of some extra moment on column. As already discussed Abaqus uses the concept of finite element method and takes membrane forces into account. So the results obtained by Abaqus are considered to be the most precise and close to reality. The results section shows how the values of different parameters differ when found out by the 3 methods. If we very carefully observe the contour diagram of the frame, we can also find the neutral axis of the frame, which passes through the center of the beam. As in theory we have studied that stress is very less at the neutral axis. This can be seen by the contour diagram also where it is clearly shown that the stress is as low as about 1 N/mm^2. Also at about 400 mm distance from the beam-column junction, we can see a blue shaded region on both sides. This region shows the point of contraflexure about which the sign of moment changes.

A very important observation from the output of Abaqus is that the maximum stress doesn't come at the junction which we expect to be by analytical methods. But instead the maximum stress value is seen at about 100mm distance from the beam-column junction. This can be very well observed by the contour diagram indicating the stress values at different points of the frame. As we know, the more meshing we do i.e. the more we divide the model into number of elements, the more precise results we will get. So at the places where we want to know the values of any variable, we should provide more and more elements. So the shifting of the position of maximum stress can be due to less meshing at the junction. The other reason can be membrane forces.

As membrane forces changes the load properties of the beam and column, so the designing of both beam and column must be revised based on the values obtained by Abaqus. This will result in more economical and more durable design as the beam will be designed according to the increased strength due to membrane forces and also the column will be designed taking the extra moment due to membrane forces into account. We must also keep in mind that the end restraining conditions of the beam influences the membrane forces. If the ends are restrained by a very stiff member then the membrane forces developed in the beam will be even higher. Earlier studies have shown that membrane forces greatly depends upon some other factors as well, like column axial stress has a major effect on the mode of failure of the frame as it increases the load needed to cause cracking and hence increases the column stiffness. Increase in the column size and column reinforcement ratio increased the axial stress above which the frame will fail by column failure. Increase in beam depth and width increased the membrane force and reduced the load at which the column failed. Increase in the beam length reduced the membrane force and increased the load at which the column failed. Reduction in the column length increased the stiffness of the support and increased the membrane force and reduced the load at which the column failed. So all these factors should also be taken into consideration if we revise our designing procedure.

CONCLUSIONS AND FURTHER DEVELOPMENTS

We can conclude that present design procedure neglects the action of membrane forces, which makes the frame uneconomical and also more unsafe and undurable. Also the deformation pattern shown by conventional frame analysis doesn't support the true deformation pattern as well as stress levels. Hence in this light, we should revise our design procedure by including the action of membrane forces into it. Or else we must use Abaqus and designing of the beams and columns must be done based on the results of Abaqus, which will result in an economical and a safe design. Or another way can be to get back to our serviceability approach, which will make the design over safe. But the best way would be to come out with a new approach of designing of members, which accounts for membrane forces as well.

REFERENCES

W. Beeby, F. Fathibitaraf. Membrane effect in the reinforced concrete frames- A proposal for change in the design of frames structures. Engineering Structures. 23(1): 82-93.

S. Timoshenko and D.H. Young. Elements of Strength of Materials. 5th Edition. Affiliated East-West Press Pvt. Ltd, New Delhi.

Abaqus User's Manual. Version 6.8. SIMULIA India-303, V.C.C., Plot No.1, LSC, Near Gulabi Bagh-Sindoran Kalan- New Delhi-110052.

STAADPro 2003 User's Manual. Research India Ltd., Bently Systems, E2-4, Block GP, Sector-V, Salt Lake, Kolkata-700 091.

IS: 456-2000. Code of Practice for Plain and Reinforced Concrete. Bureau of Indian standards, New Delhi.