7. SIMPLIFIED ANALYSIS AND DESIGN OF COMPOSITE FRAMES

7.1 INTRODUCTION

The process of structural design starts with the structural modelling followed by the analysis-design cycle with the aim of ensuring that the structure is economically able to satisfy the design requirements. The exact analysis and design of a composite building frame is a long and tedious process involving a number of complicated calculations and successive steps. The analysis is conducted on a frame model based on the assumptions including those for the structural model, the geometric behaviour of the structure and of its members and joints. The frame consists of the following members:

- Composite beam constituting of steel profile and an effective width of slab.
- Composite column which may be concrete filled tube, partially encased section or fully encased section.

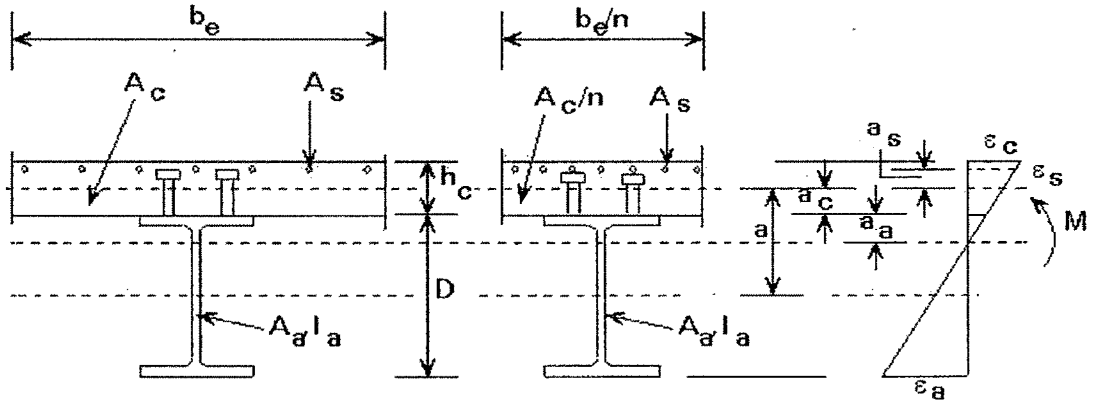
In concrete-filled tube, the steel increases the strength of the concrete because of its confining effect, the concrete inhibits local buckling of the steel, and the concrete formwork can be omitted. In encased sections, the concrete delays failure by local buckling and acts as fire proofing while the steel provides substantial residual gravity load-carrying capacity after the concrete fails.

Compared to the well known stiffness method of analysis, the moment distribution procedure yields bending moment directly, thus it avoids the tedious procedure of first finding the joint displacements and then calculating the moments. The absence of the need to solve simultaneous equations makes the method most popular. The analysis by moment distribution, however, does not usually take into account shear and axial deformations.

In the present work, the concept of equivalent stiffness is used for composite steel-concrete members and the analysis is carried out using moment distribution method. A program in the form of Excel sheet is developed to facilitate the analysis based on moment distribution method. To validate the proposed approach, the results obtained are compared by solving the same problem using the commercially available ETABS and ANSYS software.

7.2 Elastic Design Approach

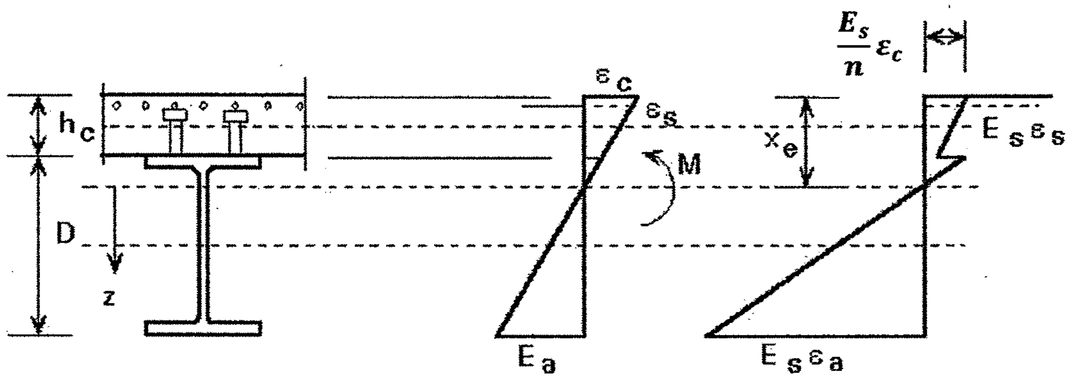
The mechanical and geometrical properties of a composite section are required for the calculation of the internal stresses and deformations. In an elastic design approach, the concrete in compression and the steel in tension are assumed to behave in a linearly elastic fashion. Where Eurocode 4 [7] permits the use of the uncracked flexural stiffness, EI_1, the concrete in tension may be considered uncracked. Where the flexural stiffness of the cracked section, EI_2, must be used, the strength of concrete in tension is ignored. Even after cracking has occurred, the section derives stiffness from the concrete. This "tension stiffening" is due to the uncracked concrete between cracks. This effect is not taken into account in the calculation of section stiffness in the present work. It is, however, taken into account indirectly in the calculation of deflections and crack widths.



a) Composite Section b) Equivalent Transformed Section c) Strain in Transformed Section

Fig. 7.1 Composite Steel-Concrete Section and Equivalent Transformed Section

In calculating the section properties of the composite section in the elastic range, use is made of the concept of the transformed section in which the steel-concrete composite section is replaced by an equivalent homogeneous section of steel. For a section subjected to positive bending, the concrete flange of area A_c is replaced with a fictitious steel flange of area A_c/n, where n is the modular ratio. The fictitious steel flange is of similar depth to that of concrete flange; see **Fig. 7.1.** Geometrical properties are readily calculated for the transformed section, and strains may be obtained using the elastic modulus for steel. Use is again made of the modular ratio in calculating the elastic stresses in the concrete flange of the original composite section as shown in **Fig. 7.2.**



a) Section b) Strain diagram c) Stress diagram

Fig. 7.2 Elastic Strains and Stresses in the Composite Section

7.3 Material Modelling

7.3.1 Elastic Modulus for Steel

As per the Eurocode 4 [7] the elastic modulus of structural steel is 210 x 10^3 N/mm^2. For both structural and reinforcing steel the linear elastic isotropic material model is considered having the elastic modulus of 210 x 10^3 N/mm^2.

7.3.2 Elastic Modulus for Concrete

Concrete is a non-linear, non-elastic material. It does not display a unique or constant value of elastic modulus as shown in **Fig. 7.3** and sustains permanent deformation on removal of load. When subjected to a constant stress, concrete strain increases with time - a phenomenon known as creep. It is also subjected to change of volume caused by shrinkage (or swelling) and temperature change.

Notwithstanding this non-linearity, it is necessary to quantify the relationship between stress and strain in order to obtain a realistic estimate of deformations. Various elastic moduli shown in **Fig. 7.3** are as follows:

- An initial tangent modulus;
- A tangent modulus corresponding to a given stress level;
- A secant modulus; and
- A "chord" modulus.

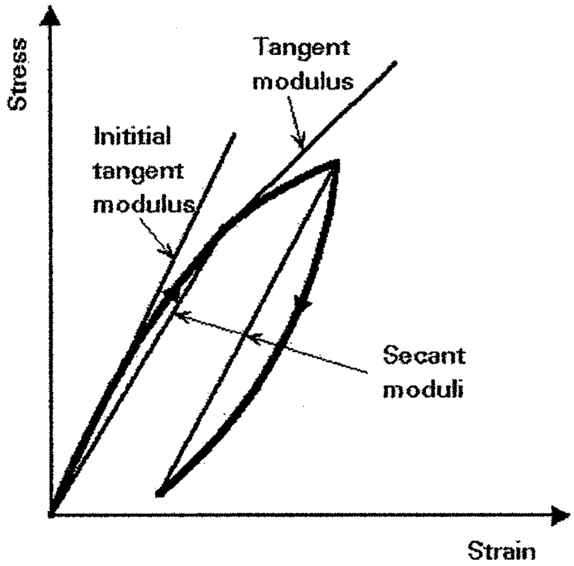


Fig. 7.3 Stress- Strain Curve of Concrete with Various Modulus

The values of a number of this modulus are seen to depend on the reference stress level. They are, in addition, affected by the rate of loading. The value used in design codes is generally a secant modulus corresponding to a specified rate of loading. An estimate of the mean value of the secant modulus (E_{cm}) for short-term loading, for normal-weight concretes, can be obtained from **Table 7.1** for the range of characteristic concrete strengths normally used in composite construction.

Table 7.1 Secant Modulus E_{cm} for Short Term Loading

Compressive Strength for	N/mm²						
- cylinder	f_{ck}	20	25	30	35	40	45
- cube	f_{ck}	25	30	37	45	50	55
Associated Mean Tensile Strength	f_{ctm}	2.2	2.6	2.9	3.2	3.5	3.8
Secant Modulus of Elasticity	E_{cm}	29	30.5	32	33.5	35	36

In the calculation of the geometrical properties of the section and stresses, reference is made to the modular ratio n. It is the ratio of E_a/E_c, where E_a is the elastic modulus of structural steel, and E_c is that of the concrete. The effect of the modular ratio on stress is illustrated in **Fig. 7.2.** For the calculation of long-term effects in buildings, sufficiently accurate results can be obtained by using an effective modulus for concrete (E_c) in the calculation of the modular ratio. The effective modulus is the short-term modulus for concrete modified for the effects of creep. Eurocode 4 [7] gives three sets of values for short-term and long-term modular ratios. These values are listed in order of increasing simplicity in **Table 7.2.**

Table 7.2 Values of the Modular Ratio

Type	Short-term effects	Long-term effects	Comments
(a)	Secant modulus E_{cm} **(Table 7.1)**	Various-depending on concrete grade	This method takes account of concrete grade and age.
(b)	6	18	Takes no account of concrete grade, but of concrete age.
(c)	15	15	Takes no account of concrete grade or age.

7.4 Sway and Non-Sway Frames

The term non-sway frame is applicable when the frame response to in-plane horizontal forces is sufficiently stiff for being acceptable to neglect any additional forces or moments arising from horizontal displacements of its nodes. The global second-order effects (i.e. the P-Δ sway effects) may be neglected for a non-sway frame. When the global second-order effects are not negligible, the frame is said to be a sway frame.

Normally a frame with bracing is likely to be classified as non-sway, while an unbraced frame is likely to be classified as sway. However, it is important to note that it is theoretically possible for an unbraced frame to be classified as non-sway (this is often the case of one storey portal frame building) while a frame with bracing may be classified as sway (possible for multi-storey building). When a frame is classified as non-sway, a first-order analysis may be used.

When a frame is classified as sway, a second-order analysis shall be used. A procedure involving iterations on a first-order elastic analysis is usually adequate for this purpose. Furthermore, if the structure meets certain conditions, a first-order analysis (without any iteration process) may be used either by making a nominal correction to member end forces to allow for the global second-order effects or by analyzing for vertical loads and for sway load effects (to be magnified for design) separately. It should be noted that bracing systems which are themselves frames (or sub frames) must also be classified as sway or non-sway.

7.5 FRAME ANALYSIS USING ETABS SOFTWARE

7.5.1 SECTION PROPERTIES

The section properties are reported in ETABS [97] software with respect to the section local axes. Also, it is assumed that the entire section is transformed into an equivalent area of the specified base material. In other words, each infinitesimal area of the section, dA, is multiplied by the ratio E_{shape}/E_{base} when computing the section properties. Using this transformation the following relationship holds true.

$$\sum_{shape=1}^{n} A_{shape}\, E_{shape} = A_{section} E_{base}$$

where, $A_{section}$ = Area reported for the section, A_{shape} = Area of a geometric shape (not reinforcing shape) included in the section, E_{base} = Modulus of elasticity of the base material, E_{shape} = Modulus of elasticity of the material specified for the shape, and n = Number of geometric shapes included in the section.

The section properties are based on the gross area of all geometric shapes transformed to an equivalent area of the base material.

7.5.2 MODEL OF COMPOSITE FRAME

Figure 7.4 shows a model of composite frame with composite encased steel concrete column and concrete beam. Model is developed in professional ETABS software. In the drawing area composite section is drawn and then its properties are defined. The frame is analysed to obtain end moments and reactions for sixteen different cases.

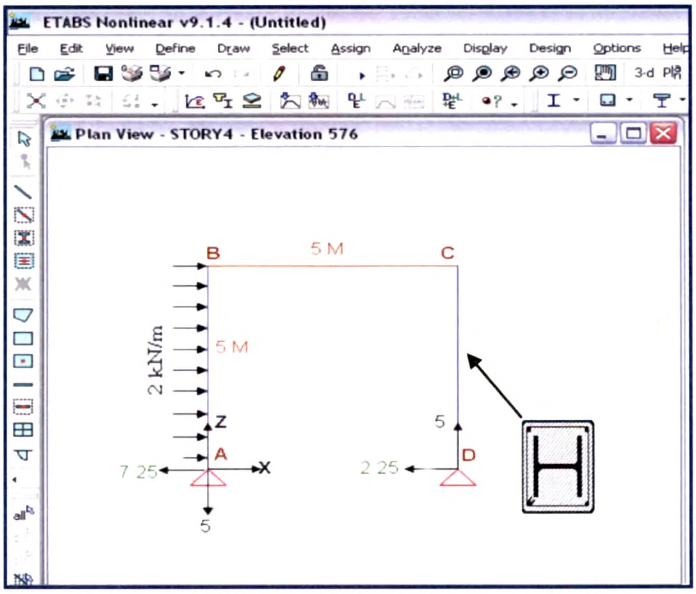


Fig. 7.4 Model of Composite Frame Developed in ETABS Software

7.6 Frame Analysis Using Moment Distribution Method

The method of moment distribution belongs to the group of approximate methods. Based on moment distribution method a general program is developed in Excel for the analysis of composite frames with due provision for various types of loadings and different types of composite sections. In case of portal frames, if the amount of 'sway' or joint movement is not known, the analysis is done by assuming some arbitrary fixed moments. These fixed moments due to side sway are then distributed and the reactions are found. The algebraic sum of horizontal reaction due to assumed sway moments must be equal to the sway force. If not, the assumed sway moment is modified proportionately.

	AB	BA	BC	CB	CD	DC
DF	0	0.407	0.593	0.593	0.407	0
FEM	-4.167	4.167	0	0	0	0
BAL	4.167	0	0	0	0	0
CO	0	2.0835	0	0	0	0
M	0	6.2505	0	0	0	0
D-1	0	-2.544	-3.707	0	0	0
CO-1	0	0	0	-1.854	0	0
D-2	0	0	0	1.099	0.755	0
CO-2	0	0	0.55	0	0	0
D-3	0	-0.224	-0.326	0	0	0
FM	0	3.4825	-3.483	-0.755	0.755	0
REACTION						
DUE TO M	-0.6965					0.151
DUE TO LOAD	5					0
TOTAL	4.3035					0.151
TOTAL SWAY	5.8475					
M	0	-5	0	0	-5	0
D-1	0	2.035	2.965	2.965	2.035	0
CO-1	0	0	1.483	1.483	0	0
D-2	0	-0.604	-0.879	-0.879	-0.604	0
CO-2	0	0	-0.44	-0.44	0	0
D-3	0	0.179	0.261	0.261	0.179	0
FM	0	-3.39	3.39	3.39	-3.39	0
REACTION						
DUE TO M	0.678					0.678
DUE TO LOAD	0					0
TOTAL	0.678					0.678
TOTAL SWAY	1.356					
MOM	0	-3.39	3.39	3.39	-3.39	0
ACT. SWAY M	0	-14.619	14.619	14.619	-14.619	0
MOM DUE TO	0	3.4825	-3.483	-0.755	0.755	0
FINAL MOM	0	-11.1365	11.136	13.864	-13.864	0
HOR. REACT	-7.2273					-2.7728
VER. REACT	-5					5

Fig. 7.5 Excel Sheet for Composite Frame

Various composite frames like composite columns with concrete beam, with or without considering self weight are analysed. Even frames having composite column with composite beam are also analysed. The screen shot of a program which is developed in Excel considering composite fully encased column with concrete beam, using the equation of equivalent stiffness of members for the frame structure, is shown in **Fig. 7.5** [99].

7.7 Frame Analysis Using ANSYS Software

7.7.1 Element Type

The ANSYS [98] element library contains more than 100 different types of elements. Each element type has a unique number and a prefix that identifies the element category: BEAM4, PLANE77, SOLID96 etc. For example, BEAM4, which has six structural degrees of freedom (UX, UY, UZ, ROTX, ROTY, ROTZ), is a line element and can be modelled in 3-D space. Here the frame has been modelled using the BEAM3 element. The element has three degree of freedom at each node i.e. translation in the X- and Y- directions and rotation about the Z-direction (**Figs. 7.6 and 7.7**). Eeq and Aeq has been provided as section properties.

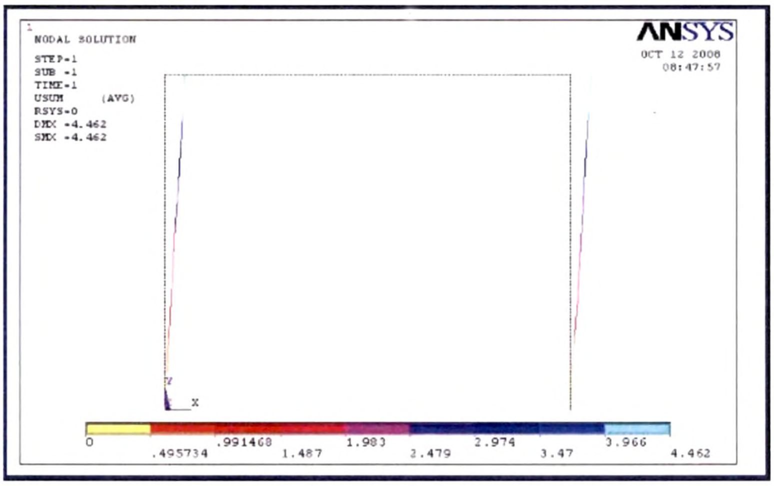


Fig. 7.6 Frame with Hinged Support having Sway Loading

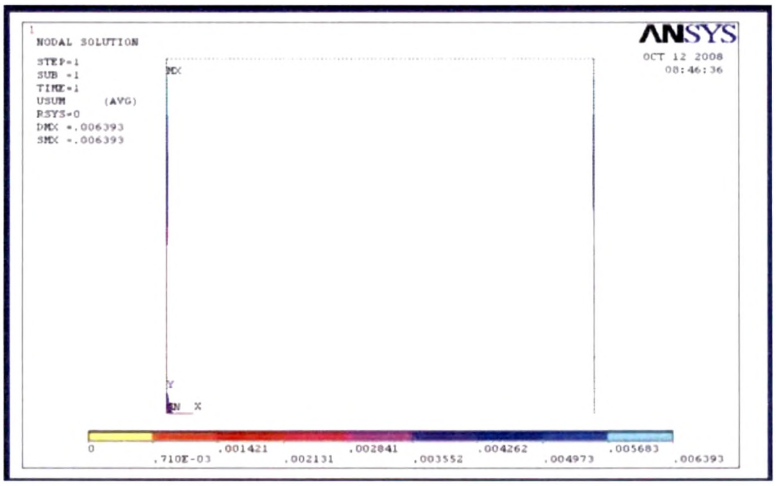


Fig 7.7 Frame with Fixed Condition with Non-sway Loading

7.7.2 MATERIAL MODELLING

Most of the element types require material properties and depending on the application, material properties may be linear or nonlinear, isotropic, orthotropic, or anisotropic, constant temperature or temperature-dependent. Linear material properties can be constant or temperature-dependent and isotropic or orthotropic. Nonlinear material properties are usually tabular data, such as plasticity data, creep data, swelling data, hyperelastic material data, etc. and anisotropic elastic material properties are usually input in the form of a matrix. These properties are different from anisotropic plasticity which requires different stress-strain curves in different directions. The isotropic linear elasticity has been considered here for modelling of the composite section. For sixteen different conditions the analysis is carried out and compared. Results are found within 4% of analysis. **Table 7.3** shows the comparison of the left hand horizontal reaction for all the cases.

Table 7.3 Comparison of Results for Horizontal Reaction

Loading	Type of Section	ETABS Result (kN)	ANSYS Result (kN)	M.D.M. Result (kN)
Sway Only	Fully encased	7.26	7.24	7.22
	Partially encased	7.14	7.145	7.12
	Pipe	7.40	7.41	7.38
	Tubular	7.20	7.20	7.24
Gravity & Sway (excluding Self Weight)	Fully encased	6.71	6.72	6.74
	Partially encased	7.05	7.04	7.02
	Pipe	6.04	6.00	5.99
	Tubular	6.98	6.8	6.46
Self Weight & Sway only	Fully encased	5.81	5.79	5.78
	Partially encased	6.54	6.40	6.43
	Pipe	4.38	4.20	4.10
	Tubular	5.33	5.2	5.16
Composite Beam Only	Fully encased	7.04	7.04	7.03
	Partially encased	6.97	6.96	6.97
	Pipe	7.00	7.10	7.09
	Tubular	7.21	7.20	7.21