Ci

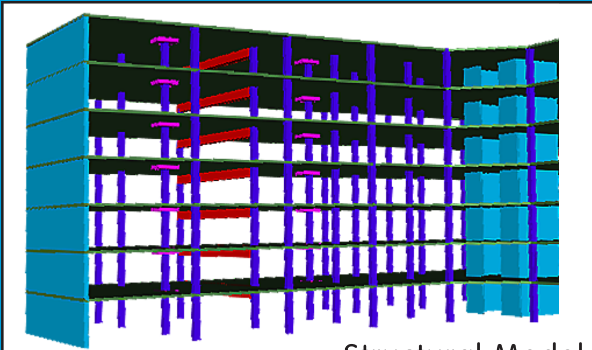
DECEMBER 2001 Vol.23 No.12

Concrete international

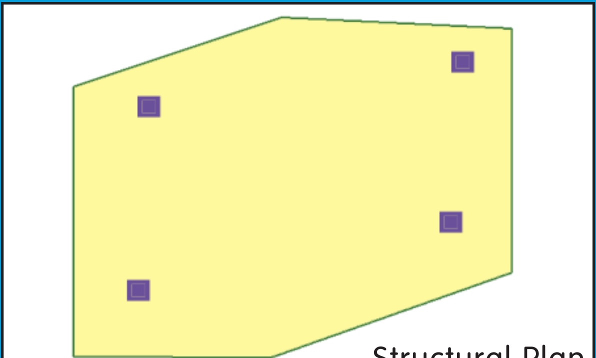
The Magazine of the American Concrete Institute - an international technical society

Computer Design of Concrete Buildings

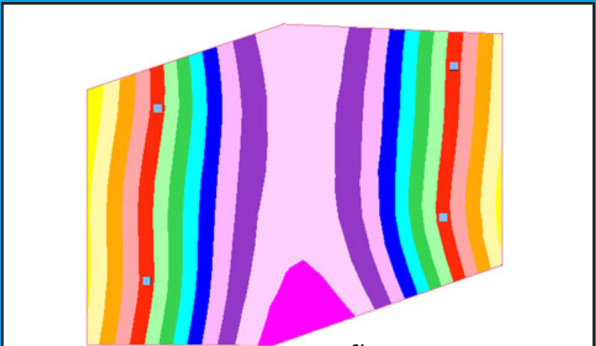
–page 28



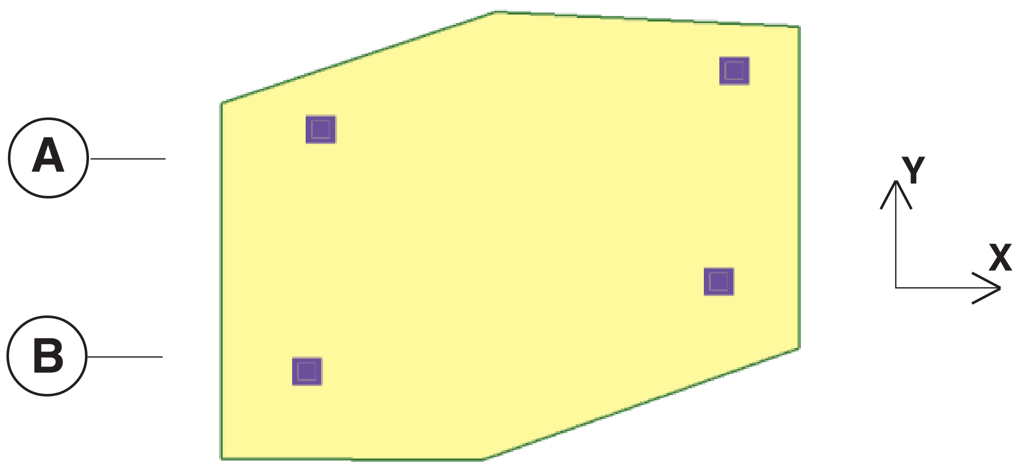
Structural Model



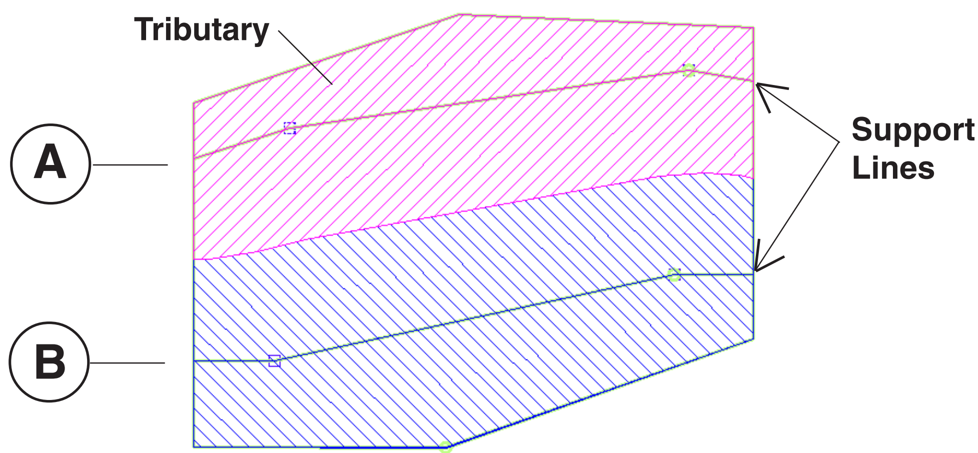
Structural Plan



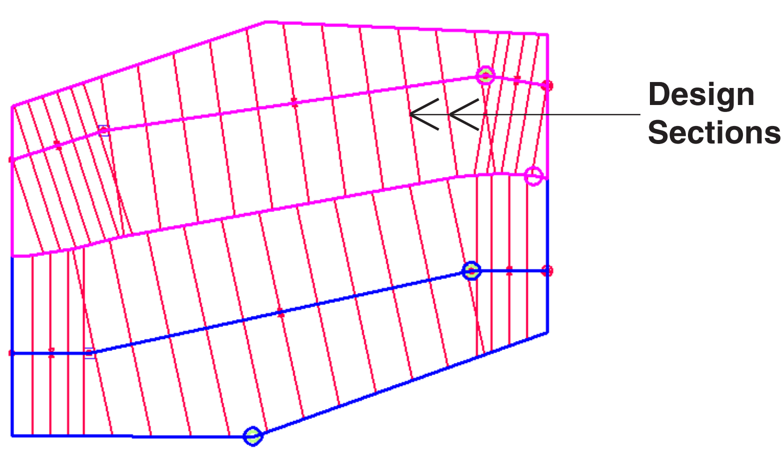
Deflection Contour



Structural Plan



Support Lines and Tributaries



Design Sections

Software for the *Design* of Concrete Buildings

Using a combination of finite-element and strip methods is emerging as the basis for automated design of concrete buildings

BY BIJAN O. AALAMI

While software for the design of steel structures provides an integrated building analysis, design, and detailing package, there is currently no comparable product for the concrete industry. This is primarily because steel buildings consist of skeletal (frame) members, such as columns, beams and girders, whereas the two major components of reinforced concrete structures are concrete floor slabs and walls, which are continuum solids. It has not been practical to develop satisfactory software that includes both gravity and wind/seismic design of an entire concrete building as a production tool for use in structural design. Recent developments in modeling techniques of concrete structures, however, coupled with advances in software technology are helping to overcome this shortcoming.

Current software technology

The commercially available design software for concrete buildings falls into the following three categories:

- Software for the *design of members* such as columns, beams, slabs, footings, and retaining walls.
- Software for the *analysis and design of floor systems*. These include beam- or column-supported one-way and two-way floor systems under gravity loading. Some floor system software can account for the effects of wind and seismic forces, if these are determined separately and entered as input data for combination with the gravity loads.
- Software for the *lateral analysis of building frames* under wind or earthquake forces. These software programs are generally limited to the design of members designated to resist the lateral loads. They don't account for the gravity design of floor slabs.

Usually, a design engineer must navigate among the three categories of software to complete the design of a concrete building. Apart from being cumbersome, extracting data from the output of one software program and for use as input in a different program is prone to error. The disjointed design process also hinders effective optimization of the design. Emerging software technology for concrete design is expected to overcome this shortcoming and provide an integrated total solution for the entire building.

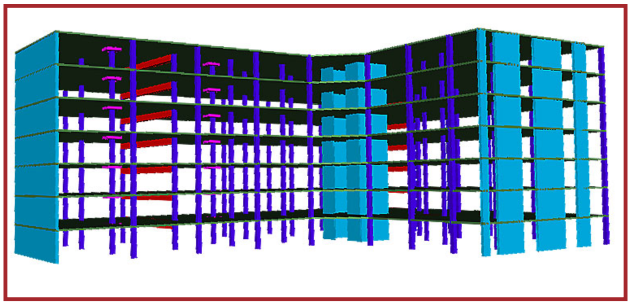


Fig. 1: Structural model of a concrete building

Automated Design Approaches

Creating the structural model

The first step in the automated design of a concrete building is structural modeling, the creation of a three-dimensional solid model (Fig. 1). The 3D structural model represents all the parts of the building that are intended to resist gravity and lateral loads. Several schemes are currently available for 3D structural modeling and visualization of the resulting building. Component technology builds the model using structural components, such as beams, columns, slab regions, and openings as opposed to using nodes and finite elements.

Element Selection

Beam elements have been used satisfactorily to represent columns and beams that are part of a concrete frame. For walls and slabs, especially the latter, shell elements are more suited than plate elements because shell elements represent both bending and in-plane (membrane) actions in a slab. Selection of plate elements as opposed to shell elements limits the analysis capability to the gravity design of flat slabs with no post-tensioning.

Gravity design of floors

Currently, gravity design of floor systems is modeled on a floor-by-floor basis. Models include all of the structural features of the floor, such as drop capitals or panels, beams, openings and steps above and below the slab. Available software permits gravity loading to be applied on the slab in any configuration, location, and value.

Wind and seismic design

General-purpose finite element programs tailored for the lateral design of buildings calculate design values for wind or seismic forces. But, none of the available finite element software includes gravity design of the floor slabs. Instead most finite element programs require rigid floor diaphragms or beam frames.

Analysis methods

Either a strip method, the Equivalent Frame Method (EFM) or the Finite Element Method (FEM), are commonly used for determining moments and shearing forces for a floor system. In practically all instances, reinforcement location and orientation are specified by the design engineer, independent of the analysis method. This decision is made on the basis of support layout, ease of construction, and code stipulations. The analysis determines the amount of reinforcement, not its location.

A combination of FEM and strip methods is emerging as the basis for automated design of concrete buildings. Principal steps in analysis and design using FEM include selection of elements, meshing and idealization of the concrete structure features. Ideally, an integrated scheme for the design of the entire building is followed by automated generation of structural drawings, so it is critical to faithfully model the actual geometry of a building as much as is practical. Otherwise, drawings generated from the structural design will show discrepancies with the architectural features and dimensions. Ideally, the structural model should include no simplifications and should represent the exact dimensions of a building.

Meshing

Concrete design is based on the integral of the actions over a design section, as opposed to the intensity of an action at a point. This permits concrete floors to be modeled using a relatively coarse mesh without loss in accuracy. The important factor is the use of automatic and well-balanced meshing with due consideration to geometry features of the structure, such as thickness changes. A well distributed meshing, about eight elements per span length, will yield acceptable designs. The following is a brief explanation of the underlying reasons.

Sensitivity of Design Values to Fineness of Meshing: Figure 2 shows the plan view of a slab supported on exterior walls and central columns under loading. Two meshing alternatives are selected. One is a coarse subdivision of the slab (Fig. 2a with 303 nodes including walls and columns), and the other a dense meshing with fine grading around the interior columns (Fig. 2b with 1285 nodes including walls and columns). The objective is to calculate the required reinforcing along gridlines 2 and 4 (Fig. 3).

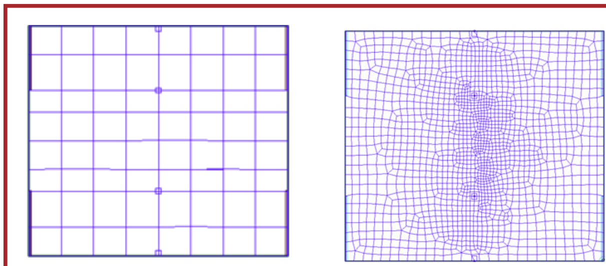


Fig. 2: Meshing alternatives for a FEM model of a floor slab supported on exterior walls and central columns

Using the fine mesh alternative, the distribution of moment about the Y-axis at the face of central supports and at the midspan is shown for the entire width of the slab (Fig. 3a) and for the tributaries of lines 2 and 4. The corresponding distributions for the coarse meshing are illustrated in Figure 3b. Note that while the shape of the distribution in the two cases is somewhat different, the

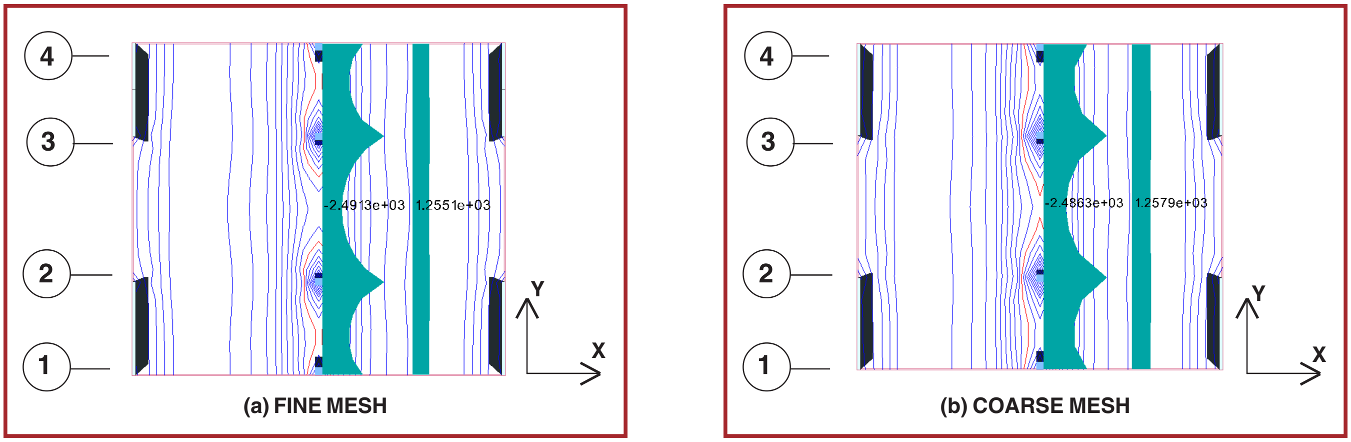


Fig. 3: Distribution of moment about the *y*-axis at the face of the central supports and at the midspan using fine mesh and coarse mesh

values needed for the design of reinforcement are almost identical for the two cases.

The design values will be essentially the same in both cases. The values of maximum deflection obtained from the coarse and fine mesh are somewhat different, but the difference is not significant enough to impact the outcome of design.

Defining the Mesh: Modern FEM software for concrete building design boasts automatic mesh generation. But there are a number of features specific to concrete design that require more accurate methods of mesh definition to capture all the geometrical features of a concrete floor.

Consider, for instance, the wall at a slab opening detailed in Figure 4a. In the traditional FEM, the wall-slab joint is modeled with a common node as illustrated in Figure 4b. While this modeling scheme gives acceptable values for design, it has two shortcomings. First, the model is not accurate enough to be used for the generation of the structural and fabrication drawings, since the slab outline doesn't agree with the architectural drawings. This leads to inaccurate calculation of bar lengths and their layout, if done automatically by the software. Second, the cross-sectional area resisting the design actions doesn't represent the actual geometry of the slab. This impacts the calculation of the minimum reinforcement, as well as reinforcement required for strength.

An improvement in this modeling is to extend the slab beyond the wall centerline to the outside face of the wall as shown in Figure 4c. In this modeling scheme, the slab is extended to the physical limit at the outer face of the wall. Combine this with the selection of a substitute analysis node that is rigidly linked to the natural nodes of the wall and slab as indicated in the figure. A similar scenario applies to the end, edge, and corner columns, where the slab extends to the outer face of the column.

Concrete floor slabs

Prior to the detailed design of a concrete floor, the engineer defines support lines and associated tributary areas. This is known as the design section. Figure 5 illustrates the floor plan of a building in which a support

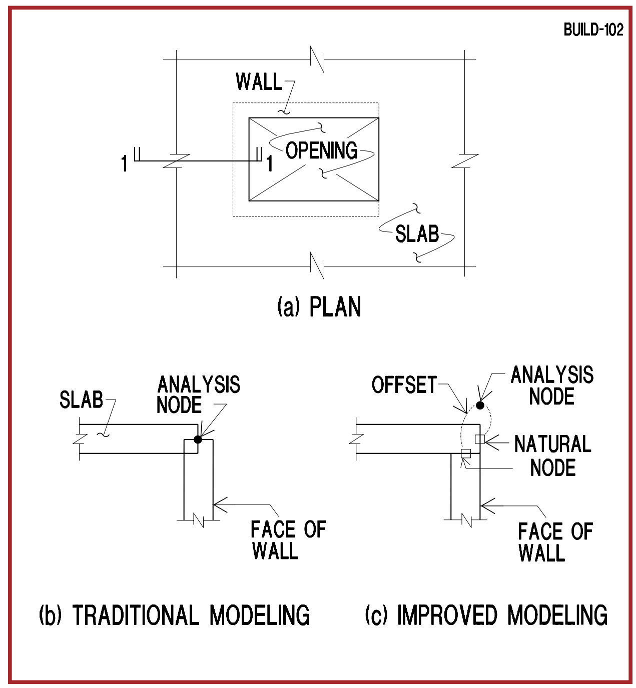


Fig. 4: Wall at a slab opening

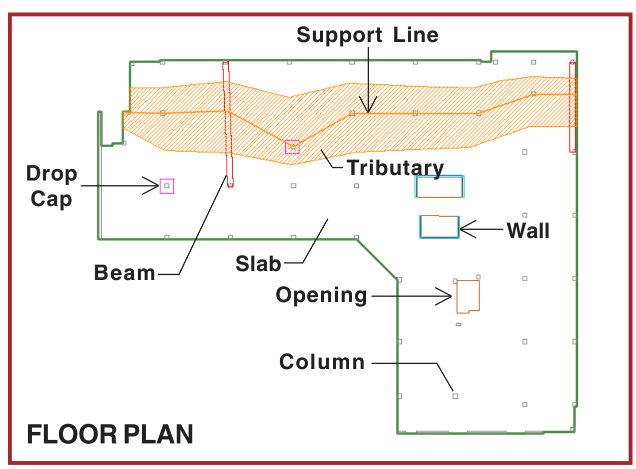


Fig. 5: Floor plan of a building defining a support line along a row of columns and its associated tributary areas

line along a row of columns and its associated tributary areas are identified. Selecting support lines and tributaries is independent of the analysis method, but a prerequisite of design. When using computer software the engineer defines the support lines and the software can automatically determine the associated tributaries. In generating the tributaries, each part of the floor is assigned to one of the support lines.

Based on the design section for each slab region, design moments and shears along a support line are determined at selected points. Typically, for each span 8 to 12 points provide a good approximation for the distribution of the design actions. At each point a section normal to the support line is drawn and the moments and shears at that section are integrated to give the design action.

The design moments are applied to the cross-sectional geometry of the associated design sections to determinate the required reinforcement If the slab is post-tensioned, the number, location and the angle of each tendon crossing the section will have to be accounted for when calculating the supplementary reinforcement.

Figure 6 illustrates the pitfall in the potential misuse of general purpose FEM design software for design of concrete floors.

The structure is subjected to a point load Q as shown. In designing section 1-1, the moment and axial force at this section are required. A coarse FEM mesh (Fig. 6b) is used. The concept of stress distribution using FEM is illustrated by examining one of the elements (Fig. 6c). For illustration, only the forces in the X direction are shown. Two observations can be made.

- The nodal forces are in equilibrium. The force P at node I is in static equilibrium with those at nodes J and K.

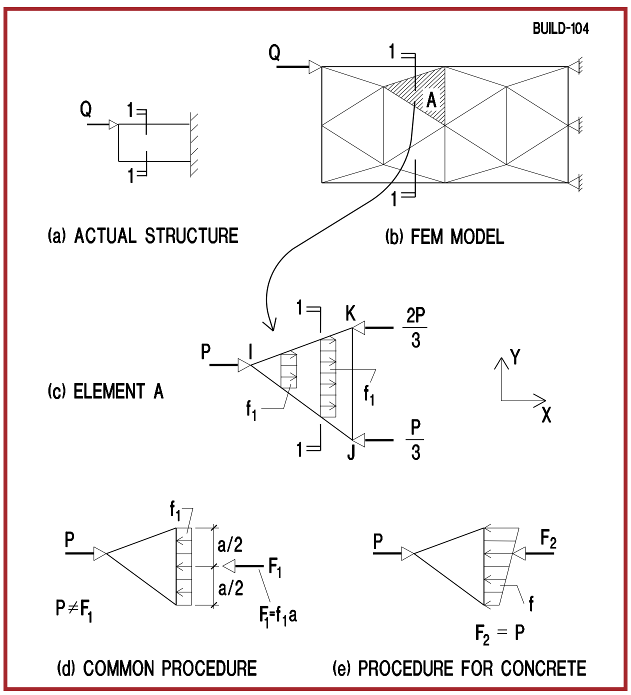


Fig. 6: Pitfall in the potential misuse of general purpose FEM design software when designing concrete floors

This is a feature of FEM formulation, and is independent of the element shape and size.

- The distribution of stress (f_1) within the element, however, is a function of the element formulation used. In its simplest form, the plane stress problem assumes a constant stress (f_1) throughout the element.

Two methods can be used to determine the contribution of the force from this element (A) toward the integrated action at section 1-1. These are shown in Figures 6d and 6e.

- In Figure 6d, the contributing force F_1 is determined from the integration of the stress f_1 over the cut length "a." This method is used almost universally in general purpose finite element programs.
- The alternative is to determine the contributing force F_2 (Fig. 6e) from the equilibrium consideration of the nodal forces (node I). In this case, from equilibrium $F_2 = P$.

The first method, which is used in the general purpose FEM, is best for problems where stress (response) at a **point** is sought. But to obtain a reliable solution, it is necessary to use a fine mesh at the point of interest. In concrete structures, however, the designer is interested in the **resultant** of the **actions** at a given section. Hence, the second method is adequate.

Beams

Most FEM, in particular software that uses plate elements for slabs assumes the beam to be centered about the mid-plane of the slab. Beams should be modeled with their centroid at the correct elevation with respect to the centroid of the slab. This is a central consideration in design of post-tensioned floors, since it is eccentricity of the tendon with respect to the centroid of the section that provides the beneficial moment used in design. Recognition of two different centroids, one for the slab and the other for the beam, at the slab beam intersection adds complexity to the modeling. Again, this complexity can be overcome through the introduction of analysis nodes and offsets as shown in Fig. 7.

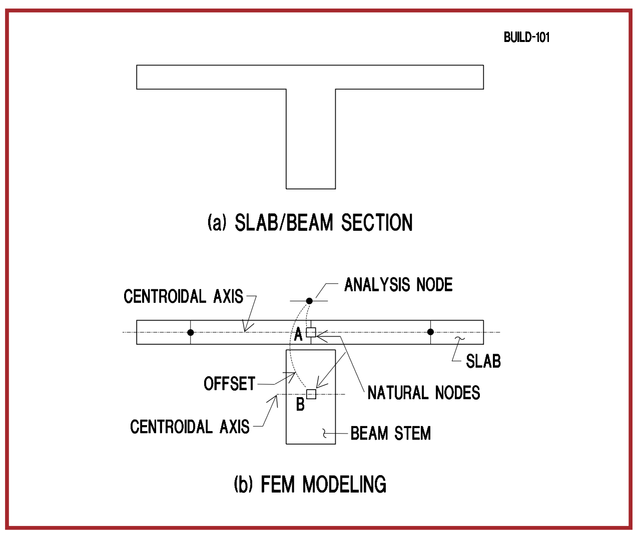


Fig. 7: Modeling of a beam/slab connection using nodes and offsets

Applied loads

Loads, regardless of the configuration of the area they cover, or the point of application should be applied accurately to the structural model irrespective of the method of analysis used. The transfer of loads to support lines, the structural components, such as beams, or finite element nodes, should take place automatically.

Computer modeling of post-tensioned floors

There are two considerations critical to successful modeling of post-tensioned floors. The first is a faithful modeling of the geometry, where the eccentricity of the tendon is maintained throughout the structure. Second, for the analysis to be capable of correctly handling most conditions, the tendon itself needs to be discretized into finite elements.

Tendon Eccentricity

Figure 8 is a view of a distributed tendon layout within two spans of different thickness. For clarity of display, support conditions and other details are omitted. A modeling scheme with shortcomings, but common in software, is to substitute a tendon with its uplift force as shown schematically in Figure 9a for the tendons displayed in Figure 8. The uplift calculated is then applied as loading to a floor modeled with plate elements (Fig. 9b). The compression due to post-tensioning is assumed to result in a uniform stress, which is subsequently added by the designer to the stresses obtained from the bending analysis. The basic shortcoming of this approximation is failure to recognize the shift in the centroidal axis of the member and the moment generated at this location by the post-tensioning force.

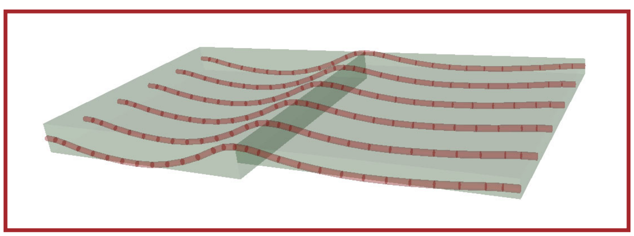


Fig. 8: A view of a distributed tendon layout within two spans of different thickness

Figure 9c illustrates the lack of alignment in the tendons at the step from the precompression forces. An effective method to overcome this shortcoming is the use of shell elements combined with representation of slab nodes (natural nodes) through substitute nodes (analysis nodes) on a reference plane (Fig. 9d). The reference plane is a fictitious plane generally assumed to be parallel to the floor slab. The natural nodes of the slab elements are mapped on this plane for analysis. At the completion of the analysis, the solution obtained is transformed to the natural nodes of each element.

Tendon Discretization

Post-tensioning tendons can be modeled either as an applied loading, or as a resisting element. The latter modeling can account for variable force along a tendon and implicit accounting for the time-dependent variables of creep, shrinkage, relaxation in prestressing and aging of concrete. The following describes the concept.

Figure 10a is partial view of a slab with a tendon. In the FEM modeling of this slab the tendon is idealized as a straight segment within each element. The shell element used for the slab models the concrete and a segment of the tendons (Fig. 10b). Each tendon segment will be regarded as a finite element. It will be initialized at the start of the calculation with the force generated at the transfer of prestressing. Subsequent changes in concrete volume due to shrinkage and creep reduce the tendon force.

Interpreting FEM analysis results

Significance of Moment Results

Most commercially available software for concrete design calculates the reinforcement on the basis of moments within each finite element. Figure 11, shows the partial view of a beam and slab with two slab elements that act as the flange of the beam. It also shows the moment (M) and axial forces (P) obtained for each of the beam and slab elements.

- When using the method of reinforcement calculation at **element level**, moment in each element is applied to the section of that element. For example, M_1 is applied to the section of element 1 to obtain the reinforcement A_{s1} as shown in part (a-ii) of the figure. The axial load P_1 acting at the centroidal axis 1 is generally disregarded. Thus

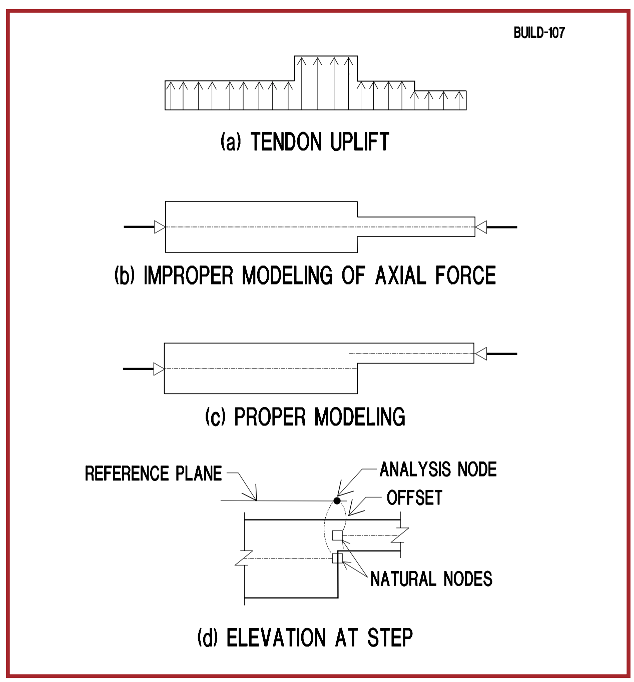


Fig. 7: Modeling of a beam/slab connection using nodes and offsets

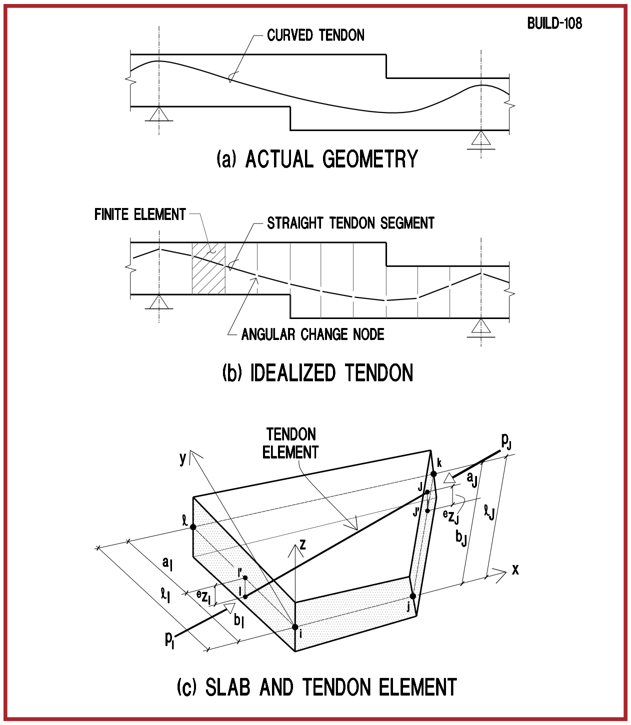


Fig. 10: FEM model of a slab with tendon

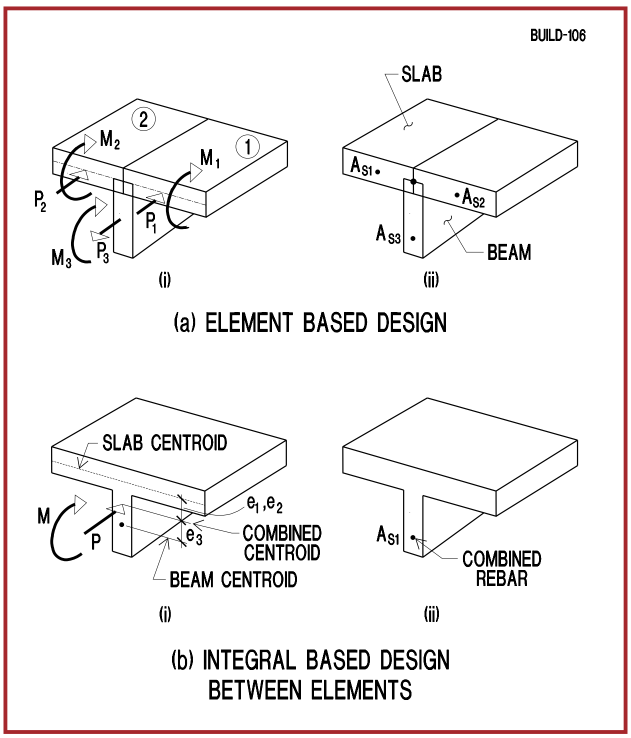


Fig. 11: Comparison between design section and element based designs

the reinforcement reported consists of intensities A_{s1}, A_{s2} and A_{s3} associated with the three elements.

- The alternative method is to calculate the reinforcement for a **design section**. In this method, for the elements 1, 2 and the beam stem, first the resultant of the three sets of element actions shown in part (a-i) is calculated. This will result in a single moment (M) and a single force (P) as shown in part (b-i). It is important to note that the moment M, is not the sum of the three moment components M_1, M_2 and M_3. The axial loads P_1, P_2 and P_3 when brought to the centroid of the combined elements 1, 2 and the beam stem result in an additional moment. The reinforcement calculated for the section is A_s and is reported at the bottom of the beam stem.

It is apparent that the second method based on **section design** is similar to the common practice of concrete design assuming the beam and slab are integral.

Stress Contours

Most finite element software is formulated to calculate the maximum surface stresses at a point. These stresses are of little significance in practical concrete design. Non-prestressed concrete floors are subject to cracking and the calculated stresses at the point of interest generally exceed well beyond the cracking limit of the concrete. In post-tensioned floors, the hypothetical stresses calculated for crack control are based on tributary actions applied to the tributary's cross-sectional area. They do not refer to a point on the concrete surface.

Deflection Contours

Deflection contours are representative of the results needed by the design engineer. Most finite element software used in practical design is based on linear elastic material properties. As such, these solutions do not account for long-term effects such as creep, nor do they account for cracking in concrete. In building construction creep typically magnifies the elastically calculated deflection by three. A multiplier between 3.5 to 4 times the computer results may be used for a realistic estimate of the long-term deflection

Challenges for the design professional

In traditional design, engineers follow a load path by clearly identifying and treating, in isolation, each structural member. Slabs, beams, girders, and columns are treated one after the other, each transferring its load to the next member. Relying on their knowledge and experience, engineers can readily evaluate the integrity of each member in isolation. With the increased use of integrated analysis and design packages, the design engineer is relieved of the tedious, step-by-step, numeric verification of the traditional method. But evaluating and validating a computer-generated design, in its entirety, is a formidable challenge.

One must first determine whether the software is applicable to the concept of design. In doing so, the engineer should be familiar with the design steps involved if no software were used. Next, the engineer should become familiar with the underlying assumptions of the software selected and determine its applicability. For example, software that doesn't represent beams in correct elevation with respect to the centroid of slab isn't suitable for post-tensioning design.

As a rule, a program should be used only if engineers can predict the general deflection and distribution of moments in the structure prior to obtaining a solution. The computed solution is used to verify the results previously predicted by the engineers. If the solution is significantly different from the their prediction, engineers should use the results only if they can satisfactorily explain the reason for the discrepancy and find it acceptable.

References

Aalami, B.O., 2000, “Strucutral Modeling of Post-Tensioned Members,” *ASCE Structural Journal,* Feb., pp. 157-162.

Aalami, B.O., and Kelley, G.S., 2001, “Design of Concrete Floors with Particular Reference to Post-Tensioning,” T*echnical Note* 11, Post-Tensioning Institute, Phoenix, Ariz., Jan., pp 16.

ACI Committee 318, 1999, “Building Code Requirements for Structural Concrete, (ACI 318-99) and Commentary (318R-99),” American Concrete Institute, Farmington Hills, Mich., 391 pp.

ACI Committee 423, 1999, “Recommendations for Concrete Members Prestressed with Unbonded Tendons (ACI 423.3R-96),” American Concrete Institute, Farmington Hills, Mich., 19 pp.

IBC, 2000, *International Building Code*, International Code Council, Inc., Falls Church, Va., 756 pp.

Selected for reader interest by the editors.

ACI member Bijan O. Aalami is emeritus professor of civil engineering at San Francisco State University and principal of the ADAPT Corp., a software and consulting firm. He obtained his PhD from Imperial College, London, and is an associate member of joint ACI-ASCE Committees 421, Design of reinforced Concrete Slabs, and 423, Prestressed Concrete. Aalami is a receipient of ACI’s Van Buren Award for application of advanced engineering to a notable structure.