Experience of design and analysis of multistory buildings

Eduard Z. Kryksunov, Mykhailo A. Mykytarenko[*]
SCAD Group, 13, Chokolovsky blrd., room 508, 03186, Kiev, Ukraine
e-mail: edk@erriu.ukrtel.net
Oleg V. Kabantsev
Central Design Institute #53, Moscow, Russia

Abstract

The report deals with issues of interconnections between different structural design models: their relationships, inheritance of data, interactions between a system and its fragments. In multistory buildings such fragments are, most often, floor panels, foundation slabs, structural cores, piers, grillages, platforms etc. The issue of correctness of fragment extraction by static and kinematical conditions is discussed, special software tools are indicated which can help in combining the analysis of a fragment and that of the whole system. It is stated that an efficient technique of source data indeterminacy consideration can be a parallel investigation of multiple competing versions of design models and the search for most disadvantageous outcomes can be performed by comparing analysis results. SCAD software suggests an automatic mode for multi-variant modeling. The report presents examples of design models created for complex structures using the SCAD software.

Keywords: finite element method, development of design models, combined response of a building and its foundation, variations of design models.

1. Introduction

The process of designing a building consists of "iterations" the final stage of which is the design itself. Though, it is neither the accurate solution (the final design of good quality) nor the number of steps (modifications of the design) nor the "lengths" of the steps (an extent to which the design is changed at each step) that are predefined. Those can be only predicted, sometimes with little or no success.

At each step one has to solve problems related to analysis and design, such as creation of design models and load models conforming to the current state of the project. Also one has to perform static and dynamic analyses, rated analyses such as wind or seismic design, other calculations such as strength of structural parts, assessment of obtained solutions etc.

2. Design models: creation, usage, modification

Modern multistory buildings are normally orthogonal in their structural solutions, in the sense that they contain chiefly horizontal (slabs, beams, trusses) or vertical (columns, walls, partitions) structural parts. Their plane outlines are pretty complicated, as a rule, with numerous elevation changes both inside and outside the buildings. A typical example is a building shown in Fig. 1.

A detailed finite element model of such building may include as many as 60,000 nodes and 60,000 finite elements or more: bars, shells, plates (including those on elastic foundations), special types etc. It is natural there are technical difficulties in creating a single design model that would both comply with all design/analysis stages and enable one to allow for all important factors.

It is characteristic for the strength analysis of multistory buildings that one uses multiple interconnected models [1] intentionally to answer the purposes of particular design phases:

(a) coarse models similar to the building being designed only in their approximate topology where it can be represented by a cantilever thin-walled bar to roughly simulate the structural core and walls of the building and to investigate integral effects such as its lowest natural frequencies or its response to constrained torsion (Fig. 2,*a*);

(b) models of the correct topology but having a coarse finite element mesh. These are needed to evaluate the interaction forces between parts of the building (loads upon the foundation of the building, forces transferred to the structural core etc.) (Fig. 2,*b*);

(c) detailed design models used to determine the design stresses in parts of the structure and check for their compliance with codes (Fig. 2,*c*).

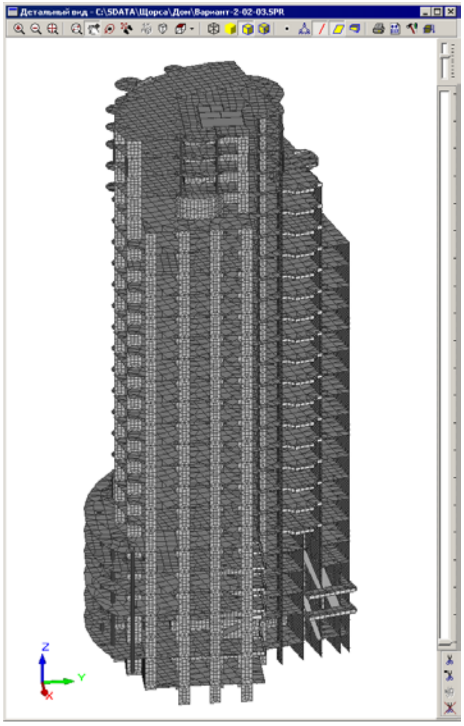


Figure 1: An example of a modern multistory building

An essential and top-priority task of the structural analysis is the gathering of loads applied to the building and determination of forces that act upon foundations and the subgrade soil. As a rule, most loads are applied at the levels of floor panels. These include the dead weight of load-carrying and enclosing parts, operational loads, loads from people in premises etc. The purpose of the load gathering and their distribution over constructions that support a floor panel is served quite well by a pretty coarse finite element mesh with its spacing about 1/3 ÷ 1/4 of the distance between supports (columns, pylons, walls and piers).

It is natural that a denser finite element mesh must be generated to determine the stress and strain distribution of the floor panel itself and to perform other design analyses (check the strength of the steel decking, proportion the reinforcement in a ferroconcrete floor). Our experience shows that in most cases it suffices to define this spacing as 1/8 ÷ 1/10 of the panel span.

Seeing that special designing (such as reinforcement proportioning in ferroconcrete parts) is needed commonly for some rather than all structural elements of a model, a design model may use finite element meshes with different spacing on objects of the same type (such as floors or walls). For example, Fig. 2,*c* shows that a denser FE mesh is used on top three floors of the building as well as on a few floors in the middle and underground parts of the structure. It is these floors that required the reinforcement to be proportioned.

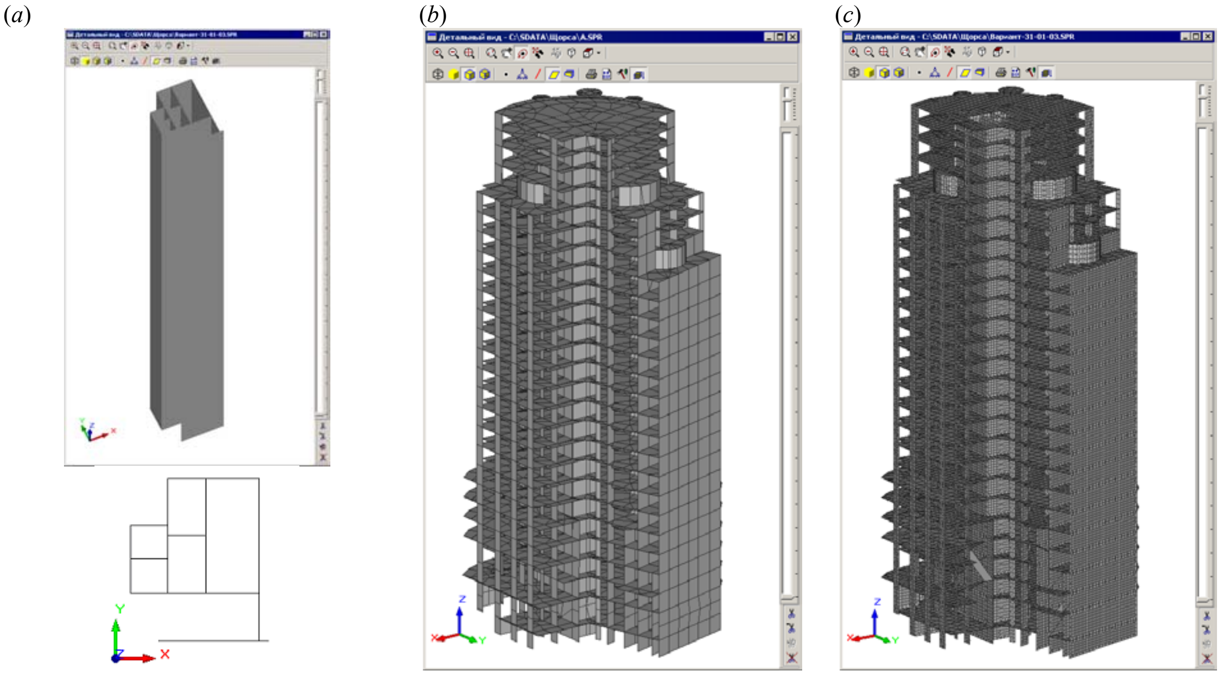


Figure 2: A set of design models: (*a*) a cantilever model (the structural core's cross-section at the bottom); (*b*) a geometrically similar model for load gathering; (*c*) a detailed finite element model for design analyses

Multiple models can be built out of a single aggregative model where there is no finite element mesh at all while all parts of the building are represented by constructive objects such as walls, floors, columns etc. [3]. This geometrical model can help generate the whole set of models automatically, from a simplest scheme intended for primary load gathering to a final design model to be analyzed in detail and to provide the compliance with codes and requirements.

The source data for the aggregative model can be the information about an object obtained from a CAD system such as architectural CAD software (AutoCAD, ALLPLAN, ArchiCAD, StruCAD, HyperStyle et al.). By using the aggregative model and automatic triangulation, one can create a finite element model of one's structure with any required density of the mesh.

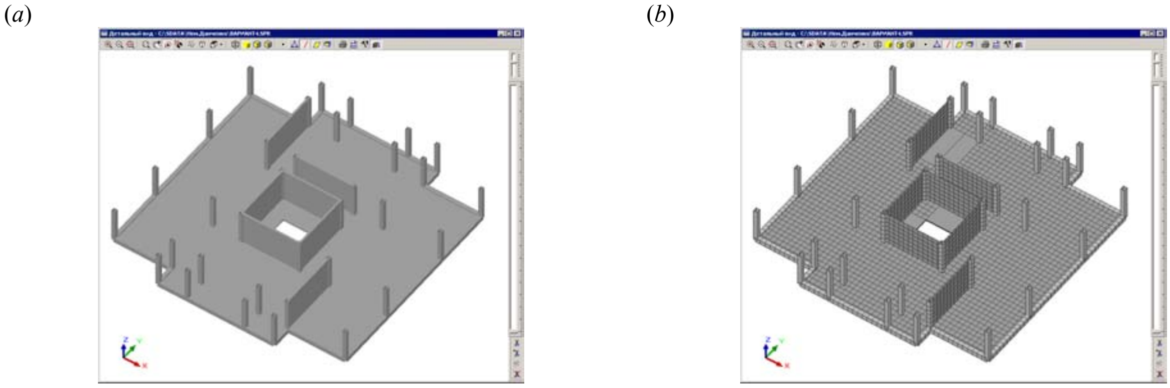


Figure 3: A part of an architectural design (*a*) and its respective finite element model (*b*)

3. A "structure-foundation" model

The existing design practice is such that first they create a general structural scheme of the whole building including its underground floors. The design of the foundation and the subgrade will be determined later on the basis of both geological features of the construction site and loads transferred from top parts of the structure.

Thus, the first phase of the design analysis is performed with two models: a geometrically correct model of the building used for the load gathering; and a model of the foundation to which the loads are applied. SCAD [4] includes special functionality for calculating loads caused by a particular part (fragment) of a structure and transferring those to another design model.

It would be more correct to have an integrated design model at once: one that would include both the building and its foundation. As a rule, such models are used at final stages of the design and analysis to refine the knowledge of the stress and strain distribution in the structure. At the initial stage of the design procedure the structure and its foundation are analyzed separately.

It should be noted that the result of the refined calculation which involves the integrated design model combined with the foundation will not differ much from the stress and strain distribution calculated by the separate models provided the soil is sufficiently homogeneous and the above-ground part of the structure is stiff enough.

In more complicated situations one may need to perform iterations to refine the stiffness and structural properties of the foundation according to this scheme: "anticipated loads upon the foundation — design model of the foundation — analysis of the above-ground structure — refined loads upon the foundation — refined model of the foundation, etc.". This procedure is actually a variation of Schwartz's iteration algorithm.

4. Model variations

It is commonly known that a design model of a structure is only an approximation of the real structure. What can be indeterminate to a great extent is the set of stiffness properties of a model. This can be due to a natural variation of properties, for example, of the elasticity modulus of concrete. Its rated value is traditionally adopted so as to comply with design codes, and its potential variability of realizations, both over time and over the object's volume, is not taken into account according to the same tradition. Though the elasticity modulus can change within a pretty narrow interval, there are numerous cases when one really has to consider the variability of the stiffness properties.

Also, joints between parts of a real structure are sometimes very far from "perfect hinges" or "perfectly rigid fixations".

Elasticity properties of natural soil beddings are widely variable. This fact makes their values highly indeterminate. The indeterminacy is contributed by both incomplete geological exploration data and approximate models of beddings and foundations. At the best, these properties may have the variance of about ± 30 %, and sometimes much greater deviations might occur.

It should be noted that there are substantial differences in the strains of elastic soil beddings caused by short-duration loads such as wind pulses or seismic actions and those caused by long-term loads such as the structure's dead weight. In the first case one uses the tangential modulus while in the second case the secant one.

There are plenty of other examples of indeterminacies that occur in design models of structural projects.

Similar cases include damages that accumulate in a structure during its life. These damages must be taken into account in analyses related to structural assessment of existing buildings, and their actual measured values should be taken into account. But some parts of an existing structure may be inaccessible for a direct inspection, therefore stiffness properties of such elements are often judged by the condition of other accessible parts. So the range of indeterminacy can be pretty wide in such cases.

An efficient method to allow for indeterminacy in source data is a parallel consideration of multiple competing versions of a design model to find the most disadvantageous solution by comparing calculation results.

The SCAD system has a special mode for processing results of analysis of multiple closely related versions of a design model. The close relationship between those should be understood as their being topologically similar, containing the same number of nodes and elements, and differing only in a few certain aspects that allow their correct comparison:

- different types of elements can be used, including "hidden" types that imitate the absence of elements while not changing their total number;
- stiffness properties of finite elements can be changed, including zero values of some rigidities and various soil reaction coefficients;
- differences are possible in the system of constraints and/or conditions of junction between elements and nodes (hinges, infinitely stiff inserts, merging of displacements).

The technique of model variation will be illustrated here by the analysis of a continuous beam lying on an elastic bed and loaded uniformly along its span (Fig. 4). In this example the response of the bed is assumed to follow Winkler's model, and the response coefficient varies within 200 t/m^3 to 500 t/m^3. Depending on the response of the bed under particular beam spans, the bending moments and shear forces vary substantially along the beam. The model variation mode enables one to choose most disadvantageous moments and shear forces for every beam's cross-section of interest. Note again that this choice is made automatically.

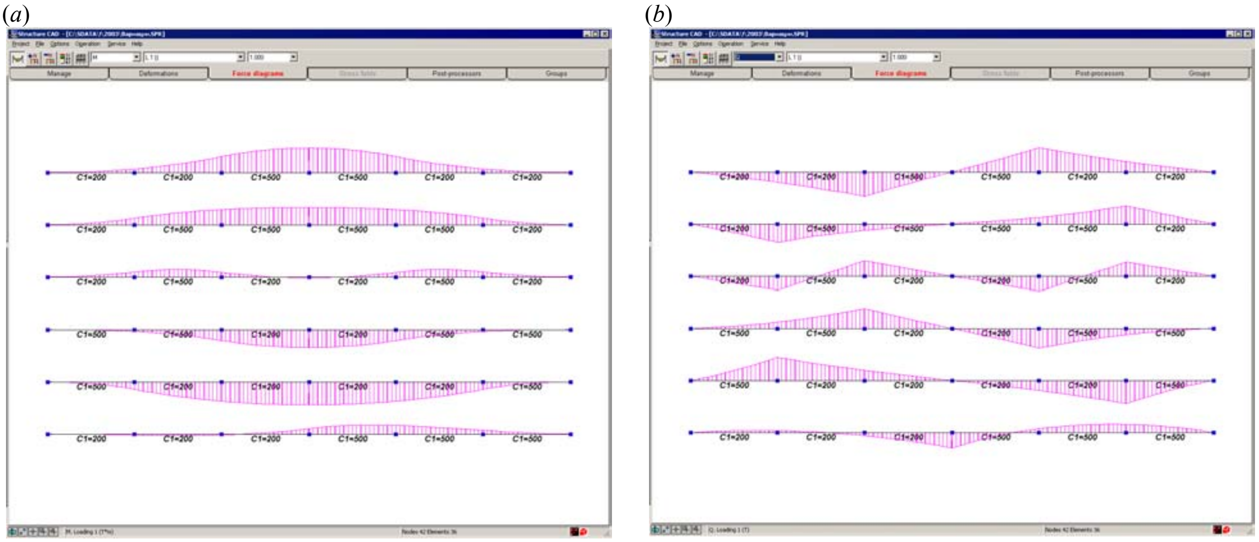


Figure 4: Results of analysis of a beam lying on an elastic bed with a variable soil reaction coefficient: (*a*) bending moments; (*b*) shear forces.

5. Dynamical models.

It is a task of great importance to investigate dynamical properties of a building's design model (periods and modes of natural vibration). As themselves, they characterize how good the quality of the design is (how successful structural solutions are) and whether the design model is correct and robust. For example, if torsion oscillation modes prevail among highest periods, then most likely the rigidities and constraints are distributed non-uniformly or otherwise improperly. Also, visualization and animation of oscillation modes help one track mistakes made in nodal joints, element junctions, rigidity properties etc.

A dynamical model is usually derived from a static one automatically by assigning inertial properties to nodal displacements and slopes. Loads upon elements and weights of those connected to nodes are taken into account.

Dynamical actions are peculiar in that the structure interacts with its loads because dynamical properties of the system define the magnitude of the loads to a great extent. An example of this kind can be our above-mentioned case of an elastic soil bed loaded by various types of loads: permanent, long-term, short-term, or other actions. Elastic properties of the bed are defined by the load type: if there's a long-term load then they are functions of the deformation modulus, if there's a short-term or other special load then those are functions of the elasticity modulus. This fact changes the character of the stress distribution substantially both in load-bearing constructions and in the soil bed.

The standard approach to the simulation of loads upon a structure without taking into account what has been said above may result in a completely distorted result. For example, the existing practice of a theoretical evaluation of dynamical properties of skeleton ("flexible") structures includes creating models that contain only load-carrying parts of the structure. Enclosing constructions and their connections to the load-carrying structures are ignored in the structural model, as a rule. In most cases a design model of a building does not include interior self-supporting walls and partitions, exterior wall enclosures though these parts possess some load-bearing capacity (chiefly due to their shear rigidity). Only their masses are taken into account in inertial properties of the model. This circumstance affects little the response to static vertical loads, but it influences very much the periods of natural oscillations and magnitudes of horizontal dynamical actions.

This approach to design modeling can be reasonable in cases when an action (being intensive enough) cuts off the said connections because of their negligible load-bearing capacity comparing to the intensity of the action. Examples can be found in the seismic structural analysis because seismic design codes allow for the possibility of damages in both nonbearing parts (partitions, self-supporting walls etc.) and in load-carrying constructions of buildings.

Under actions of far lower intensities such as wind loads (except for ultimate cases such as hurricanes or tornados) the connections between load-bearing and secondary nonbearing elements of a building are not cut off, as a rule. So the said constructions will affect the deformation of the load-bearing parts because they have connections to those. Thus they will influence the dynamical response of the structure too.

Fig. 5 shows results of experimentation [4] with dynamical properties of a building the model of which contained the variable amount of auxiliary structural elements (partitions, cantilever enclosing constructions etc.). It can be clearly seen how the natural frequencies vary as we compare the building with no partitions (Fig. 5,*a*) and that with 50% or 100% of the partitions present. It means that the presence of interior walls and partitions increases the stiffness of the building and reduces the highest natural oscillation period to a great extent.

Tests have shown that the presence of partitions in the amount that conforms to a conventional residential house increases the principal natural frequency 1.72÷2.46 times comparing to the "clear" frame. When cantilever exterior self-supporting walls are added, the frequency increases 2.75 times.

Dynamical properties of the models differently filled by nonbearing elements have been evaluated using the SCAD software. Results of this evaluation have shown a pretty high correlation with those of experimental investigations (Table 1).

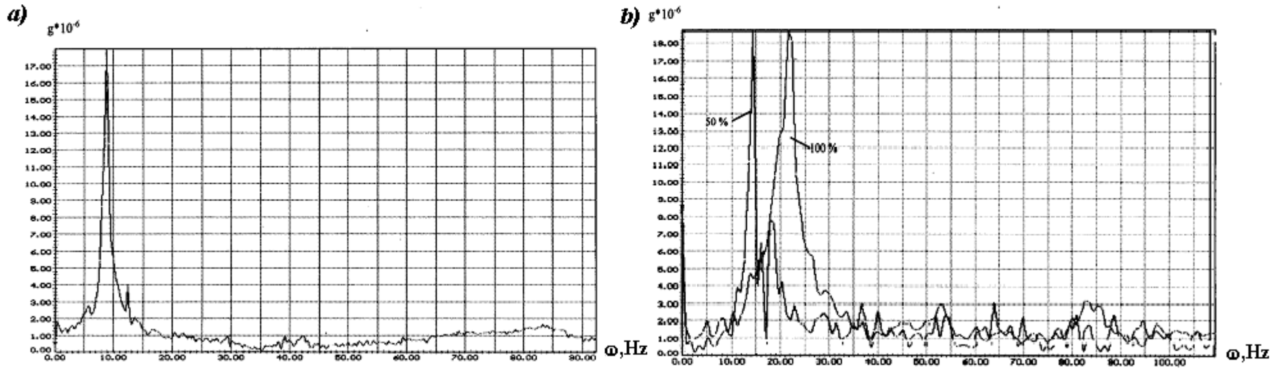


Figure 5: A structure's oscillation spectrum: (*a*) no partitions; (*b*) partially or completely mounted partitions

Table 1: Comparison of calculated and experimental dynamical properties

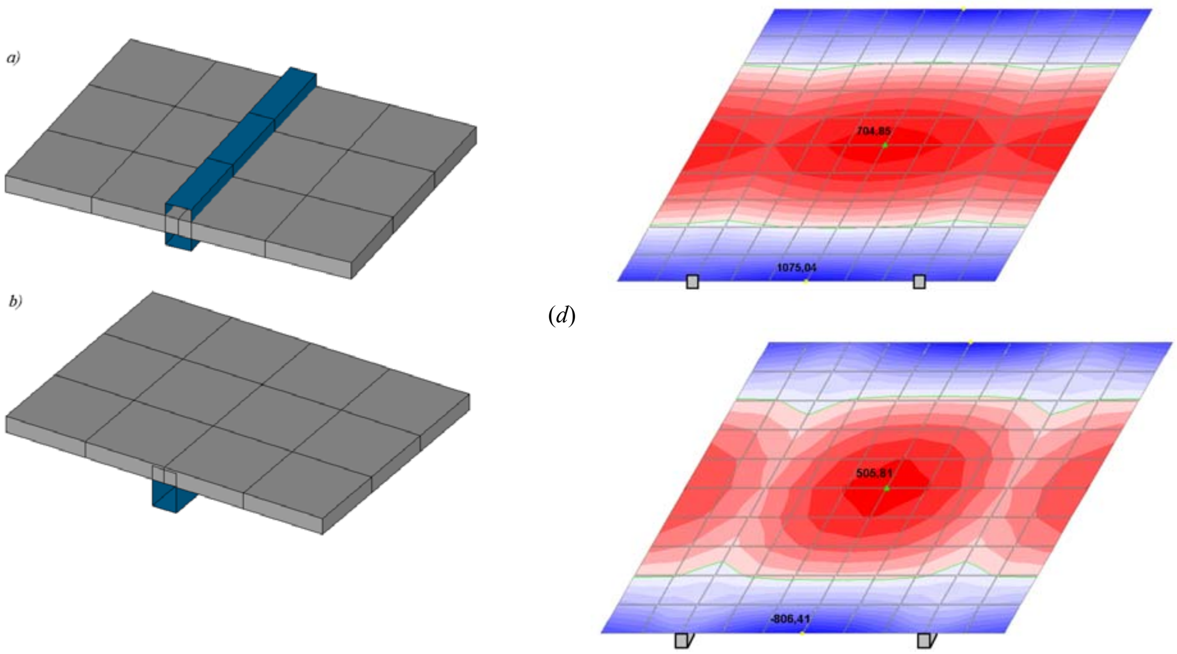
Model type	Frequency of oscillation of 1st mode, Hz	
	Calculated	Experimental
A structural framed model without nonbearing elements	9.6	9.2
A structural framed model with its 1^{st} story filled by nonbearing elements (partitions)	13.2	13.4

So it seems that multiple models are really needed in order to construct a faithful prediction of the stress and strain distribution in load-bearing parts of a structure. Each of the models should take into account both properties of the load-bearing parts themselves and interactions between the structure and loads applied to it.

6. Modeling of floors.

Let's pay some attention to local issues very important for building a correct design model. These include the modeling of floors, junctions between columns and floor panels, and the like.

As a rule, floor panels of buildings are made of monolithic ferroconcrete, and they may also contain beams arranged along contours of separate bays. The design of a floor is such that its top surface is smooth (i.e. beams do not stick out of the floor upwards). When modeling a ferroconcrete slab by plate or shell finite elements and modeling beams by bar elements, the middle surfaces of the plates should stand higher than the elastic parts of the bars.


(*c*)

(*d*)

Figure 6: Modeling a ribbed floor (a fragment): bars connected to the slab's nodes without stiff inserts (*a*) and with stiff inserts (*b*) and respective stress fields on the top surface of the slab (*c*, *d*)

Conditions of strain compatibility between the bars and the plates will be satisfied only if the bars are connected to the plates' nodes by perfectly rigid vertical inserts (Fig. 6,*b*). The membrane group of stresses arises in the plates in this design which generally results from the correct modeling of the floor.

If the bars join the plates' nodes directly with no stiff inserts then the membrane stresses do not arise in the plates under vertical loads. This modeling conforms to the case where in the "real" construction the beams as if stick out of the plates up (Fig. 6,*a*). The former model is more correct though it may take more effort to create.

The difference in stresses calculated by the models (*a*) and (*b*) is more noticeable than the difference in the reinforcement of the floor panels made after the calculated stresses (Fig. 6,*c* and 6,*d*). It can be explained by two circumstances:

— the stresses in the middle surface of the panels are compressive and are resisted by ferroconcrete with little or no help from the reinforcement;

— the reinforcement assortment is discrete and the diameter of bars used is almost always constant, therefore the difference between reinforcement required by the design and that actually used by builders is leveled.

The simulation of joints between floor panels and columns requires accuracy and carefulness in the course of the model creation. It is especially important in cases when a column joins a floor directly. In a simplest model when the whole stress is transferred to a single point, the node of junction between the column and the panel (Fig. 7,*b*), a very big stress appears in the latter (Fig. 7,*d*) which is far from real.

A simple technique shown in Fig. 7,*a* can be used to allow for the fact that the real load is transferred via a small spot which conforms to the cross-section of the column. Here a perfectly rigid body is introduced into the area of the panel coincident with the column's cross-section. Due to this no big stress appears in the junction with the column as shown in Fig. 7,*c*.

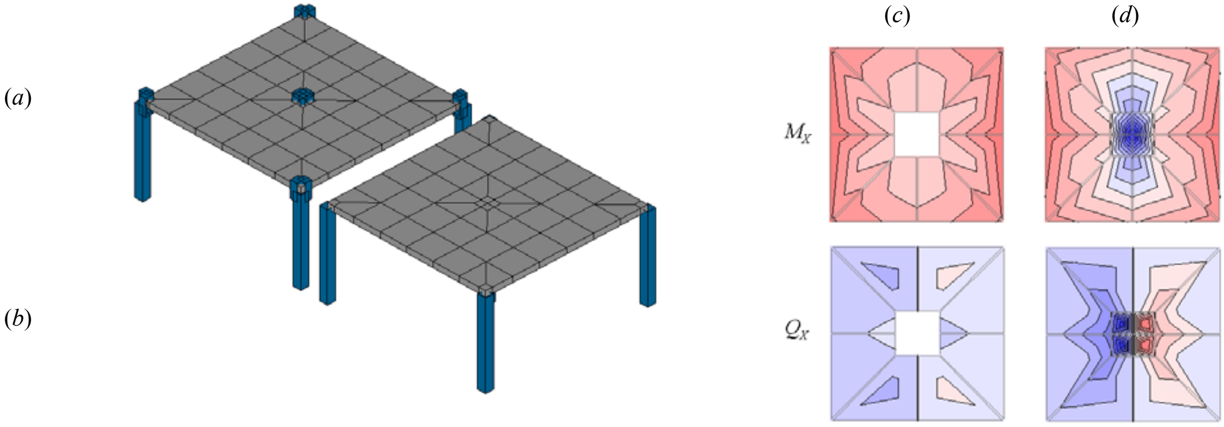


Figure 7: Modeling the area where a slab is supported by a column

7. Modeling of the soil subgrade response.

Let's discuss the soil from the viewpoint of its mechanical model. As a rule, designing a structure requires data of geological explorations in several boreholes on the construction site to be known. This information includes pointwise data regarding soil layers occurrence, their composition and mechanical properties.

The CROSS software [5] is used to determine coefficients of reaction of soil subgrade under a foundation slab. The source data that this software needs include a finite-element model of the foundation imported from the SCAD system, geological exploration data specified as properties of soil layers, and loads on the foundation slab. Also, spots of foundations of nearby structures can be specified if they are believed to affect the soil response.

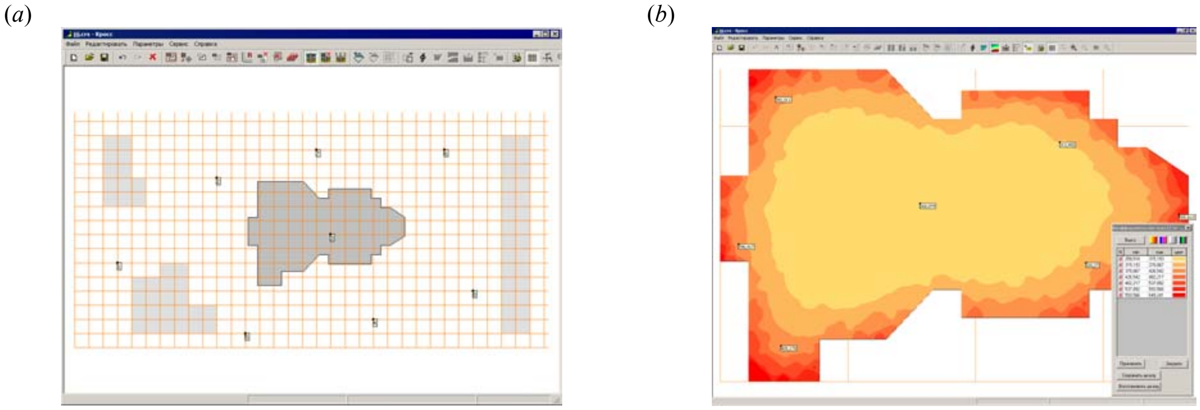


Figure 8: Calculation of soil reaction coefficients by the CROSS software: position of the foundation, holes and existing objects on the construction site (*a*) and distribution of the soil reaction coefficients under the building's spot (*b*).

On the basis of these data, the structure of the soil massif is restored and displacements of the day are determined in nodes of the finite element mesh on the spot of the structure's foundation slab. By dividing the applied loads by the foundation's surface displacements we obtain the soil subgrade reaction coefficients.

The soil reaction coefficients thus obtained enable us to automatically take into account geological features of the soil and location of nearby structures on the construction site, to serve the purpose of the stress and strain analysis of the structure.

8. Conclusion

The SCAD computer program is a core of a big software system called SCAD Office. This software package is intended for both structural strength analysis and other design activities including the execution of engineering documentation. The system includes satellite programs integrated with the main program that are intended for building sections and calculating their geometric parameters (the whole set of those can be calculated including sectorial properties; welded, solid and thin-walled sections can be designed). There are programs for designing and analyzing steel, ferroconcrete and stone structural elements in compliance with effectual design codes.

The SCAD software has been used to perform a great deal of designs and calculations of civil engineering objects. As a rule, those objects have complicated geometries and often must be simulated using interconnected finite elements of different dimensions (one-dimensional, plane, spatial) with tens of stiffness values. A lot of most various loads including dynamic loads have been taken into consideration in such analyses.

The experience of the software's applications evidences that it is really efficient in performing analyses of the discussed types.

The SCAD Office software has been in use since 1994 and found a vast field of application (over 2000 installations) in design institutions of the former USSR and in other countries of Europe, Asia and Africa.

References

[1] Perelmuter, A.V. and Slivker, V.I. *Analysis of structures — models and interpretations,* 2nd ed., Kiev, "Steel" Publ. Co., 2002, (in Russian)

[2] Kryksunov, E.Z., Zelivyanskiy, E.B. *An object-oriented graphical preprocessor FORUM* (in these Proceedings)

[3] Karpilovsky, V.S., Kryksunov, E.Z., Perelmuter, A.V. at al. *SCAD for users*. Kiev, Compass Publishing House, 2000 (in Russian)

[4] Tjnkikh, G.P., Kabantsev, O.V., Dorofeev, M.L. The experimental research of the influence of nonconstructive elements on periods of normal oscillation of frame buildings, *Earthquake Engineering. Safety of Structures*, №6, 2002, pp. 12-16.

[5] Karpilovsky, V.S., Kryksunov, E.Z., Mykytarenko, M.A. at al. *Implementation of SNiP in constructional programs*, Kiev, Compass Publishing House, 2001 (in Russian)