MECHANICS

UDC 539.42
https://doi.org/10.23947/2687-1653-2022-22-4-306-314

Original article

Strength Calculation of the Coupling of the Floor Slab and the Monolithic Reinforced Concrete Frame Column by the Finite Element Method

Peter P. Gaidzhurov ✉, **Victor A. Volodin**
Don State Technical University, 1, Gagarin Sq., Rostov-on-Don, Russian Federation
✉ gpp-161@yandex.ru

Abstract

Introduction. The given model does not allow obtaining data on the distribution of the stress tensor components in the zone of intersection of the floor and the column. Therefore, the problem of improving the strength calculation technique at the joints of floors and columns is urgent. This study aims at developing the concept of fragmentation of the frame to assess the load-bearing capacity of the floors. As a rule, a frame-rod design scheme is used under the finite element modeling of high-rise buildings made of monolithic reinforced concrete. Numerical experiments using volume-rod and volume-plate models of a repeating structural fragment were performed on a test example of a six-span three-storey monolithic reinforced concrete frame. Practical recommendations have been developed for the refined strength calculation of the floors of monolithic reinforced concrete frames of multistorey buildings.

Materials and Methods. Computational experiments were performed using the ANSYS Mechanical software package, in which the finite element method was implemented in the form of a displacement method. A plate-rod ensemble of finite elements was used to simulate the stress-strain state of a monolithic reinforced concrete frame. The refined calculation of the coupling zone of the floor slab and column under static loading was performed using solid, beam, truss and plate elements.

Results. An engineering technique has been developed for numerical analysis of the stress-strain state of the coupling of the floor and the column of the reinforced concrete monolithic frame under static loading. The most accurate result was provided by a finite element model constructed using beam finite elements as reinforcing rods.

Discussion and Conclusions. The developed technique of numerical modeling of the coupling of the floor and the column made it possible to estimate the real strength margin of this node, taking into account the real geometry of reinforcing grids, as well as to clarify the bearing capacity of a monolithic reinforced concrete frame under various loading scenarios.

Keywords: finite element method; solid, beam, truss, plate finite elements; girderless floor with a capless joint; model of coupling of a floor slab and a column; models of discrete floor reinforcement.

Acknowledgements. The authors would like to thank the reviewers, whose critical assessment of the submitted materials and suggestions contributed to a significant improvement in the quality of this article.

For citation. P. P. Gaidzhurov, V. A. Volodin. Strength Calculation of the Coupling of the Floor Slab and the Monolithic Reinforced Concrete Frame Column by the Finite Element Method. Advanced Engineering Research, 2022, vol. 22, no. 4, pp. 306–314. https://doi.org/10.23947/2687-1653-2022-22-4-306-314

Introduction. Currently, the design of high-rise buildings made of monolithic reinforced concrete is based on a frame-link structural scheme, which, to a certain extent, enables to provide the "survivability" of the building in case of progressive (avalanche-like) destruction [1, 2]. Note that in the domestic construction science, the term "progressive destruction" is understood as the process of collapse of support structures on several floors of the building or on one floor with an area of more than 80 m^2. This phenomenon occurs as a result of the simultaneous destruction, as a rule, of one support element, followed by a rapidly increasing destruction of the entire building or part of it according to the "domino" scenario.

The main structural elements of monolithic reinforced concrete frames of multistorey buildings are repeated fragments of columns and girderless floors connected by capless joints [1].

Despite many years of experience in designing buildings with braced frames, there are cases of progressive destruction of these objects in world practice. The reasons for such phenomena are mainly due to errors in the design of the interface zone of floors and columns in combination with violation of the established rules of operation of buildings. This study aims at developing an engineering technique for strength calculation of the braced frame of a multistorey building made of monolithic reinforced concrete, taking into account the volumetric nature of the stress state in the area of the floor and column.

In finite element modeling of bending reinforced concrete structures, an approach is usually used that involves the representation of concrete by two-dimensional or three-dimensional finite elements (FE), whose construction is based on the principles of elasticity theory. Reinforcing rods, as a rule, are modeled by beam or truss rods of the appropriate dimension. According to the method of ensembling solid and rod FE, the following schemes of reinforcement representation are distinguished [2, 3]: discretely distributed, in which the coordinates of nodes of different types of elements coincide (Fig. 1 *a*); reinforcement using the so-called built-in finite elements (Fig. 1 *b*). In the latter case, the coordinates of the nodes of different-type FE do not coincide, and the procedure of "condensation" of the elements of the stiffness matrices of the rod element on adjacent nodes of the solid (basic) element is specified (Fig. 1 *c*). Note that the computational technology of embedded FE is applicable exclusively for solving the problem of generalized plane stress state. Moreover, basic FE should be isoparametric and polyquadratic, and the built-in FE should be straight-line truss-type (Fig. 1 *c*).

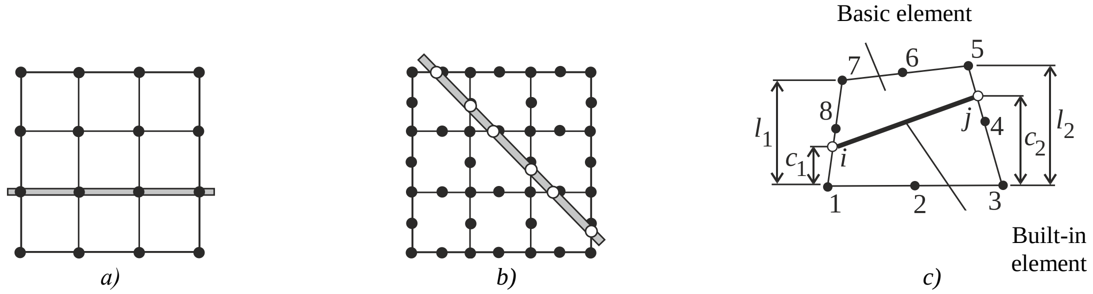


Fig. 1. Reinforcement modeling schemes: *a* — discretely distributed;

b — with built-in rod elements; *c* — basic and built-in elements (the authors' figure)

In the Russian practice of strength calculation of monolithic buildings and structures made of reinforced concrete, customized software complexes LIRA-CAD and SCAD-Office are mainly applied [4, 5]. They use the technology of the ensembling of plate and beam FE for the construction of braced frames. At that, the so-called rigid inserts are automatically introduced in the areas of the interface of floors and columns. They are star-shaped spaced beam elements with artificially high bending stiffness (Fig. 2). The geometry of the rigid insert corresponds to the transverse dimensions of the column section. This approach makes it possible to give a more physical character to the distribution of bending moments in the floor FE.

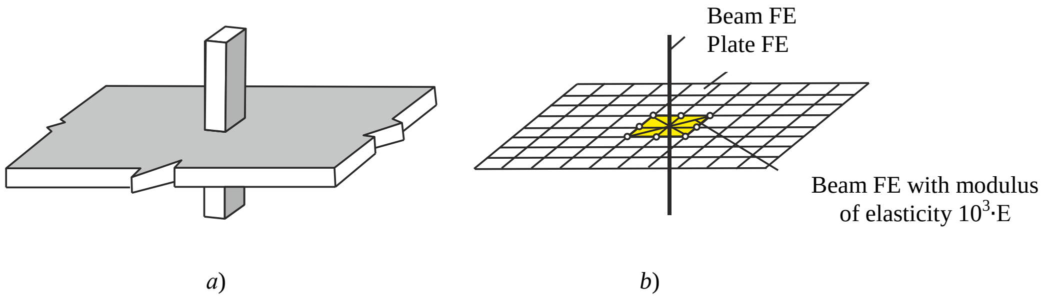


Fig. 2. Modeling of the floor and column interface zone:
a — interface zone; *b* — design scheme of the interface zone with a rigid insert (the authors' figure)

An alternative to rigid inserts with beam FE with excessive rigidity is the procedure of co-opting (binding) the degrees of freedom of the nodes of plate elements adjacent to the node of the rod element, with the corresponding nodal displacements and rotation angles of the FE rod elements. Naturally, with this approach, it is required to provide for the thickening of the grid to the dimensions of the cross sections of the columns at the junctions of the plate and rod FE. Note that the considered approach can also be programmatically automated.

Materials and Methods. At present, developers of finite element software complexes, when constructing stiffness matrices of plate and shell finite elements, widely use the MITC (Mixed Interpolation of Tensorial Components) algorithm, based on the procedure of independent (separate) approximation of bending and shear deformations. This procedure aims at eliminating the effect of "jamming" or false shift.

To model the columns of monolithic reinforced concrete frames, rectilinear two-node beam FE with six nodal dofs are used, which include three displacements in the direction of local axes and corresponding angular displacements.

The ANSYS Mechanical software package provides a four-node plate FE, SHELL63, and a two-node beam FE, BEAM188[1], for calculating spatial braced frames. The analysis of the three-dimensional stress-strain state of reinforced concrete structures was performed using a special eight-node element SOLD65.

Research Results. Let us calculate the stress-strain state of a monolithic three-storey reinforced concrete frame with a column pitch of 5×7 m. Floor height — 4.7 m; column cross-section size — 0.4×0.4 m; floor thickness — 0.2 m. Elasticity modulus, Poisson's ratio, specific gravity of floor materials — $E = 2.7\times10^4$ MPa; $\upsilon = 0.2$; $\gamma = 2{,}440$ kg/m ^{3}; columns — $E = 3\times10^4$ MPa; $\upsilon = 0.2$; $\gamma = 2{,}500$ kg/m ^{3}. The calculated values of concrete resistance to axial compression and stretching are: $R_b = 25.5$ MPa, $R_{bt} = 2.37$ MPa, respectively.

We assume that constant evenly distributed load $q = 2$ kPa acts on all the floors of the frame. Calculations are carried out taking into account the own weight of the frame. We believe that the bases of the columns of the first floor of the frame are rigidly fixed.

At the first stage of the calculation of the floor and the frame columns, we model, respectively, plate SHELL63 and beam BEAM188 FE.

Table 1 shows the results of comparative calculations of the ceiling of the first floor of the frame. Calculations were carried out with a grid step on the floors of 0.2 m and 0.5 m. The columns were divided into 6 FE in both cases. In Table 1 and further, the following is indicated: u_z — maximum deflection; M_x, M_y — bending moments relative to global axes. The third row of Table 1 shows the calculation data of the frame with rigid inserts (grid pitch 0.2 m). Rigid inserts with side dimensions of 0.4×0.4 m were modeled by SHELL63 plate FE with elasticity modulus $10^3{\cdot}E$.

Note that values $\mathrm{M}_{x\,max}$, $\mathrm{M}_{y\,max}$, given in Table 1, refer to small local zones of coupling of columns and floors. Values $\mathrm{M}_{x\,min}$ and $\mathrm{M}_{y\,min}$ are distributed along the perimeter of the frame between the rows of edge columns.

[1] Basov KA. ANSYS: spravochnik pol'zovatelya. Moscow: DMK Press; 2005. 640 p. (In Russ.)

A similar calculation of the frame was performed using LIRA-CAD software package. The value of the maximum deflection on the ground floor under similar loading and mechanical constants of the material, obtained using the LIRA-CAD software package, was u_z = –3.06 mm, which is comparable to the calculation in ANSYS u_z = –3.49 mm with a grid step on the floors of 0.2* (Table 1). When calculating using LIRA-CAD complex, at the step of constructing an "analytical model" at the points of intersection of columns and floors, "punching contours" were included, affecting indirectly the bending stiffness of the floors in the direction of its increase.

Table 1

Values of extreme bending moments in the interface zone of the first floor ceiling and column when using SHELL63

Grid pitch, m	Number of unknowns	u_z, m	M_X, kN·m		M_Y, kN·m	
			min	*max*	*min*	*max*
0.5	103,032	–0.004908	–13.1	37.2	–20.6	45.3
0.2	583,200	–0.005409	–15.2	70.2	–22.5	99.7
0.2*	583,200	–0.003492	–10.8	106	–17.5	106

Table 2 shows values of the extreme bending moments for the ceiling of the first floor, obtained using ANSYS (grid pitch 0.2* m) and LIRA-CAD (grid pitch 0.395 m). It should be noted that in LIRA-CAD complex, a scale of linear bending moments has been introduced to quantify values M_x and M_y, i.e., the resulting moments are reduced to a strip of plate with a width of 1 m.

Table 2

Comparison of extreme values of bending moments in the interface zone of the first-floor ceiling and column, calculated using ANSYS and LIRA-CAD

Software package	M_x, kN·m		M_y, kN·m	
	min	*max*	*min*	*max*
ANSYS	–10.8	106	–17.5	106
LIRA-CAD	–31.5	8.37	–42.2	12.9

From the data in Table 2, it can be seen that the extreme moments of the same name obtained using ANSYS and LIRA-CAD differ significantly in magnitude. This circumstance is explained by the presence of local zones of concentration of internal forces at the interface points of plate and rod FE in the investigated finite element model. Moreover, the accepted dimensions of the plate FE in both complexes do not allow us to accurately simulate the gradients of changes in bending moments in the specified concentration zones.

Analyzing the results of the calculation of the frame according to the plate-rod scheme, we come to the conclusion that this model does not enable to study a detailed picture of the stressed state of the structure. In particular, it is not possible to analyze the zones of tensile normal stresses σ_x, σ_y σ_z, that occur at the junctions of columns and floors. In this regard, the task of constructing a computational model of a repeating fragment of the frame is urgent. That allows performing a numerical study of the volumetric stress-strain state, including reinforcement. Figure 3 shows the floor framing plan with selected repeating fragments *a*, *b*, *c*. The area of the repeating fragment *a* is the object of further study.

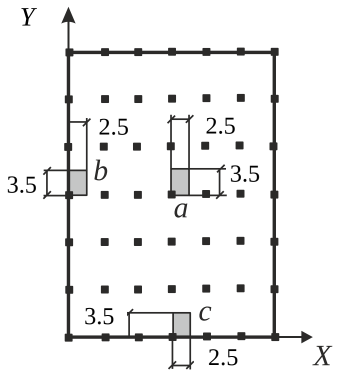


Fig. 3. Repeating fragments *a*, *b*, *c* of floor framing (the authors' figure)

To simulate the volumetric stress-strain state of a repeating fragment, we used an eight-node SOLID185 type FE with three degrees of freedom in the node.

Figure 4 shows the design scheme of the repeating fragment *a* of the frame under consideration. The above calculation scheme refers to the first floor of the frame. Here, the concentrated force *P* =72.6 kN was taken from the diagram of longitudinal forces obtained using a plate-rod model of the frame. In this case, the concentrated force *P* was converted to a statically equivalent pressure $q_к$ = 453.5 kPa, acting on a site of 0.2×0.2 m (1⁄4 part of the column section). Static boundary conditions $\bar{u}_X$, $\bar{u}_Y$, $\bar{u}_Z$ were imposed on the FE nodes with regard to the cyclic symmetry of the pattern of floor deformation. The corresponding finite element model of the fragment, built on the basis of SOLID185 solid FE, is shown in Figure 5. In this case, the grid step of the solid FE was assumed to be 0.1 m. We applied pressure on the floor fragment and 1/4 of the column using SURF154[2].

The comparison results of the stiffness properties of the plate-rod and solid finite element models of the frame fragment under consideration in the form of displacement distribution patterns u_Z are shown in Figure 6. The distribution patterns u_Z in Figure 6 *b* and Figure 6 *c* correspond to the calculations of the fragment with and without taking into account force *P*. As can be seen from the presented results, the calculation without taking into account force *P* (Fig. 6 *c*) gives the displacement values closest to the data of the plate-rod model. Further numerical experiments will be carried out taking into account the loading of the fragment column by pressure $q_к$.

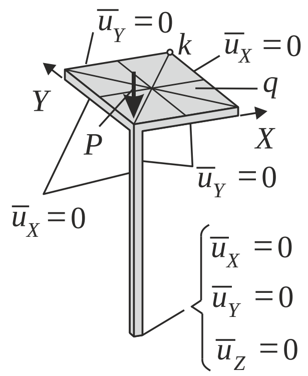


Fig. 4. Fragment calculation scheme (the authors' figure)

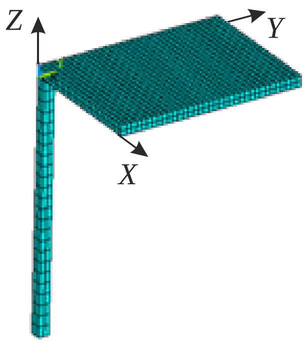


Fig. 5. Finite element fragment model (the authors' figure)

[2] Basov KA. ANSYS: spravochnik pol'zovatelya. Moscow: DMK Press; 2005. 640 p. (In Russ.)

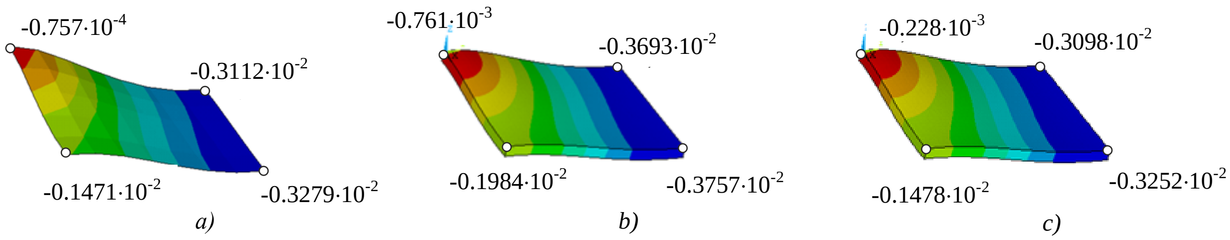


Fig. 6. Calculation results of displacement u_z for two models: *a* — plate-rod model;
b — solid model taking into account force *P*; *c* — solid model without taking into account force *P* (the authors' figure)

Let us study the effect of reinforcement on the stress-strain state of the frame fragment under consideration. We assume that the floor is girderless and capless. Let us consider three schemes for modeling the reinforcement of the part of the floor belonging to the fragment: 1 — discrete reinforcement using rod FE (BEAM188); 2 — discrete reinforcement using truss FE (LINK180); 3 — distributed reinforcement using plate FE (SHELL63). It should be noted that reinforcement modeling by means of truss FE is also suitable for calculating prestressed reinforced concrete structures[3] [1, 6, 7].

The structural elements of the discrete reinforcement of the frame fragment are shown in Figure 7. The material of the reinforcing rods is steel ($E = 2\times10^5$ MPa; $\nu = 0.28$; $\gamma = 7{,}800$ kg/m^3).

Reinforcing grids are made of rods of the following diameter: background reinforcement (along the entire floor plane) — 10 mm; reinforcement of girders — 12 mm; reinforcement of the column cap — 16 mm. The diameters of the rods for transverse reinforcement of girders and column caps have values similar to the above. The reinforcing rods of the column with a diameter of 12 mm are located at the corners of the section.

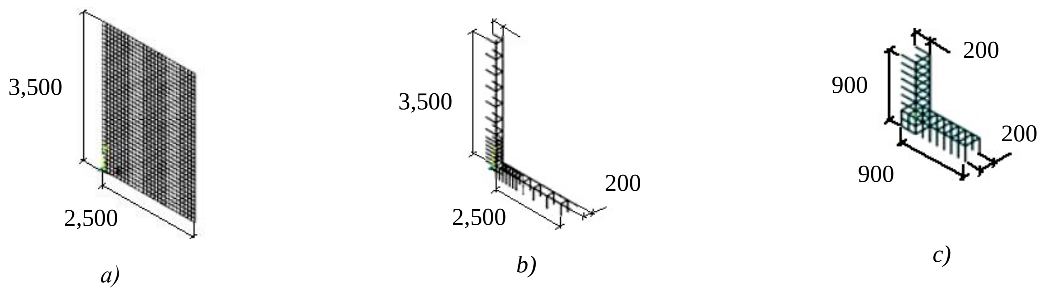
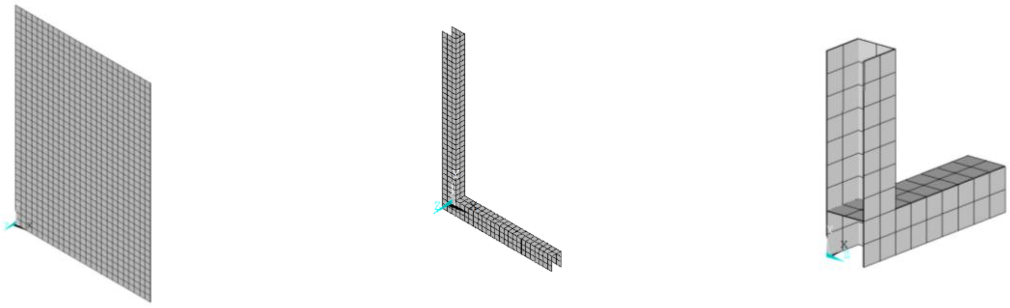


Fig. 7. Structural elements of discrete reinforcement:
a — background upper/lower reinforcement; *b* — girder reinforcement;
c — column cap reinforcement (the authors' figure)

The assigned values of the diameters of the reinforcing rods were taken on the basis of the prototype [8–10]. To model the reinforcement, we used either beam or truss FE.

An alternative to the discrete reinforcement model is an approach based on the principle of "smearing", the so-called distributed reinforcement model. In general, the principle of "smearing" is the introduction of the reduced modulus of elasticity, which is a function of the elastic moduli of the components of an inhomogeneous material and their volume concentrations. However, the calculation practice [1] has shown that for reinforced concrete structures working on bending, the introduction of specific reinforcing layers equivalent to the volume occupied by reinforcing rods gives more realistic results. The geometry of such layers mimics the geometry of reinforcing steel grids. For the frame fragment under consideration, Figure 8 shows equivalent reinforcing layers modeled by SHELL63-type plate FE.

[3] CS 52-101-2003. Concrete and Reinforced Concrete Structures without Prestressing. Moscow, 2004.



a) b) c)

Fig. 8. Equivalent reinforcing layers: *a* — layer for background reinforcement (upper/lower); *b* — girder reinforcement layers; *c* — column cap reinforcement layers (the authors' figure)

Based on the condition of equality of the volumes of reinforcing elements and the coincidence of the geometry of the reinforcement zones, we obtain the following thicknesses for the equivalent reinforcing layers: a layer of background reinforcement — 1.6 mm; layers of girder reinforcement — 1.1 mm, 0.74 mm, 1.2 mm, 0.78 mm; layers of column cap reinforcement — 5 mm, 2.7 mm.

During the computational experiment, the dependence of deflection u_z at point *k* and maximum tensile stresses $\sigma^+_{x\ max}$, $\sigma^+_{y\ max}$, $\sigma^+_{z\ max}$ on the adopted reinforcement scheme of the fragment under study was investigated (Fig. 4). Table 3 shows the calculation results.

Table 3

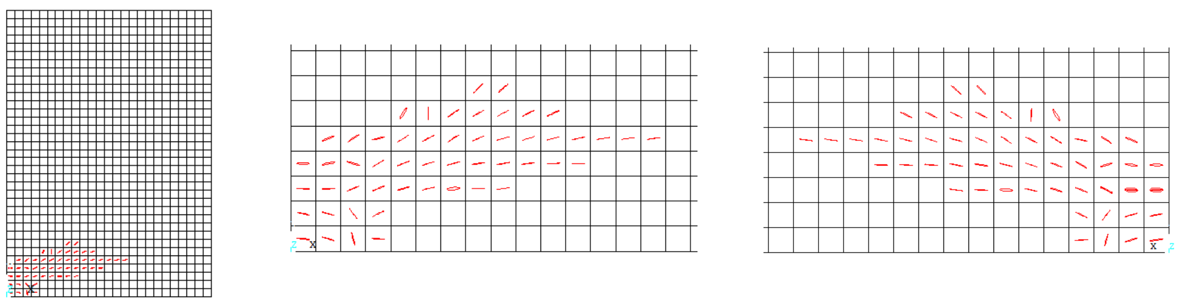
Values of deflections and stresses in various methods of reinforcement modeling

Reinforcement modeling	u_z at point *k*, mm	$\sigma^+_{x\ max}$, MPa	$\sigma^+_{y\ max}$, MPa	$\sigma^+_{z\ max}$, MPa
Nonreinforced	–3.32	3.26	4.40	1.44
BEAM188	–2.81	2.77	3.76	1.08
LINK180	–2.92	2.75	3.74	1.06
SHELL63	–2.06	1.19	1.74	0.513

The analysis of the results has shown that the use of plate FE causes a significant underestimation of the maximum tensile stresses. The data obtained using beam and truss FE are practically the same, which is explained by the low bending stiffness of the reinforcing rods.

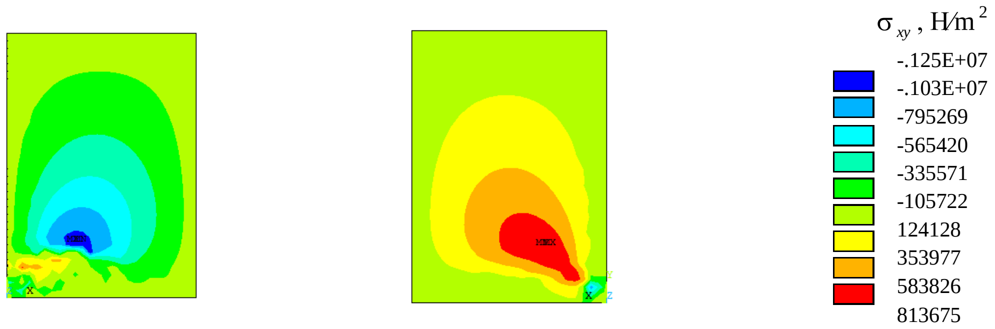
It is important to emphasize that for the considered loading option, when using rod FE (BEAM188 and LINK180), in the column – floor interface zone, the strength condition for stresses $\sigma^+_{x\ max}$ and $\sigma^+_{y\ max}$ is not fulfilled.

Figure 9 shows a visualization of the cracking process in the floor slab obtained through modeling concrete with SOLIDS65 solid eight-node FE. It should be noted that only microcracks appear in the floor, whose boundary is in good agreement with the shear stress field σ_{xy} in the floor plane (Fig. 10).



a) b) c)

Fig. 9. Visualization of cracking process in the floor slab:
a — general view of the fragment; *b* — view of the crack formation zone from above;
c — view of the crack formation zone from below (the authors' figure)



a) *b)* *c)*

Fig. 10. Stress distribution pattern in the floor:
a — view of the fragment from above; *b* — view of the fragment from below; *c* — stress scale (the authors' figure)

To simulate the physical nonlinearity of concrete, the Willam-Warnke model [11] was used with the following parameters: the coefficient of shear force transmission at an open crack — 0.3; the coefficient of shear force transmission at a closed crack — 0.7; the tensile crack reduction factor — 0.6.

Discussion and Conclusions

1. Based on the performed linear-elastic calculation of a monolithic frame made of reinforced concrete, it has been validated that the plate-rod model widely used in design practice does not allow quantifying the values of tensile normal stresses in the areas of conjugation of floor slabs and columns in the areas of interface of floor slabs and columns.

2. A technique has been developed for numerical modeling of the volumetric stress-strain state at the interface zone of floor and column of a monolithic reinforced concrete frame, which provides estimating the actual margin of safety of this node, as well as to specify the bearing capacity of the corresponding building or structure under various loading scenarios.

References

1. Frederick S. Merritt, Jonathan T. Ricketts. Building Design and Construction Handbook. McGraw Hill Professional, 2000. 1600 p.

2. Wolanski AJ. Flexural Behavior of Reinforced and Prestressed Concrete Beams Using Finite Element Analysis. Marquette University, Milwaukee; 2004. 87 p.

3. Hyo-Gyoung Kwak, Filippou FC. Finite Element Analysis of Reinforced Concrete Structures under Monotonic Loads. University of California, 1990. 124 p.

4. Gorodetskiy AS, Batrak LG, Gorodetskiy DA. Raschet i proektirovanie konstruktsii vysotnykh zdanii iz monolitno-go zhelezobetona (problemy, opyt, resheniya i rekomendatsii, komp'yuternye modeli, informatsionnye tekhnologii). Kyiv: Fakt; 2004. 106 p. (In Russ.)

5. Perelmuter AV, Slivker VL. Raschetnye modeli sooruzhenii i vozmozhnost' ikh analiza. Moscow: DMK Press; 2009. 596 p. URL: https://znanium.com/catalog/product/1910747 (accessed: 14.09.2022). (In Russ.)

6. Samokhvalova EO, Ivanov AD. Styk kolonny s bezbalochnym beskapitel'nym perekrytiem v monolitnom zdanii. Magazine of Civil Engineering. 2009;3:33–37. (In Russ.)

7. FIP Handbook on Practical Design: Examples of the Design of Concrete Structures. Thomas Telford Limited, London; 2010. 188 p.

8. Lukin AV, Modestov VS. Finite Element Modeling and an Analysis of the Stress-Strain State of Reinforced Concrete Structures. St. Petersburg Polytechnic University Journal: Physics and Mathematics. 2014;201:35–46.

9. Gaigerov PP, Al-Jabobi Sami Fahl, Al-Yaj Mahmoud Abdo Hasa. Finite Element Modeling of Force Transmission the Tension of the Steel Tendon on the Concrete. University News. North-Caucasian Region. Technical Sciences

Series. 2017;2:73–78.

10. Gaigerov PP, Iskhakova ER, Al-Jabobi Sami Fahl, et al. Definition of Camber Long-Span Reinforced Concrete Girders in pre-Stressed Reinforcement Finite Element. University news. North-Caucasian region. Technical Sciences Series. 2017;4:86–91.

11. Karpenko NI. Nauka, tekhnika i obrazovanie Obshchie modeli zhelezobetona. Moscow: Stroiizdat; 1996. 416 p. (In Russ.)

Received 31.10.2022.

Revised 22.11.2022.

Accepted 29.11.2022.

About the Authors:

Gaidzhurov, Peter P., professor of the Engineering Mechanics Department, Don State Technical University (1, Gagarin sq., Rostov-on-Don, 344003, RF), Dr.Sci. (Eng.), ORCID, gpp-161@yandex.ru

Volodin, Victor A., post-graduate student of the Engineering Mechanics Department, Don State Technical University (1, Gagarin sq., Rostov-on-Don, 344003, RF), ORCID, vik25vol@bk.ru

Claimed contributorship:

P. P. Gaidzhurov: task statement; choice of solution method; construction of mathematical and computer model; discussion of the results. V. A. Volodin: critical review of literary sources on the research topic; computational analysis; discussion of the conclusions.

Conflict of interest statement

The authors do not have any conflict of interest.

All authors have read and approved the final manuscript.