УДК 624.53(045)

Skrebnieva Dariia[13], *assistant*
National Aviation University, Kyiv, Ukraine

NUMERICAL METHODS OF CALCULATION OF STRESS-STRAIN STATE OF COMBINED ELEMENTS OF PLATING FROM PROFILED SHEETING

In the article numerical methods for calculating steel-reinforced concrete slabs with profiled sheeting are proposed, which are non-uniform composite structures, for which it is necessary to have reliable and practical methods of their calculation which take into account the heterogeneous structure of structural elements, real load schemes, boundary conditions, etc.

Keywords: steel-concrete slabs with profiled sheeting, non-uniform composite structures, numerical methods of calculation, stress-strain state, finite element method.

Formulation of the problem. Analytical methods of calculation of steel reinforced concrete slabs can be considered such that only to some extent describe the work of slabs. This is due to the fact that different assumptions or hypotheses about the nature of deformation of slabs are used to obtain finite equations. Most of these disadvantages are devoid of numerical methods based on the direct solution of the equations of the theory of elasticity. Depending on the discussed tasks, these methods can be used both for studying the stress-strain state (SSS) of slabs, and for their simplified engineering calculations.

Review of the recent sources and publications. Among the various softwares that implement the calculation of the stress-strain state by the finite element method, the most common and powerful is LIRA software. Its capabilities are good suited for scientific research, including a variety of finite element models, nonlinearities, material contact, etc. It was decided to make calculations using the finite element method in the LIRA software.

Problem statement. The carrying out of calculation and research SSS of combined steel reinforced slabs (CSRS) by the finite element method (FEM). The main purpose is to determine the order of constructing models of the CSRS, which correspond to the real work of such plane and volume (3D) of finite elements (FE), connected in each other in the nodes. Combination of different types of (FE) in one model allows significantly simplifying it without loss of accuracy of calculations, facilitating data processing and reducing the time of calculation.

At researching of slabs SSS ПМА-1, ПМА-4 і ПМА-5 the attempt of use 3D FE for modelling of profiled sheeting with the aim of researching of type FE influence on calculation accuracy.

The contact layer was modeled using volume (3D) FE. The contact layer connected 3D FE of steel reinforced concrete 2D finite element profiled sheeting.

The based material. The construction of the geometric model was performed in the software Allplan, selected from library of finished elements the necessary profiled sheeting (Figure 1). The length of the profiled sheeting corresponded to the length of the experimental samples and was equal to 1500 mm.

A similar approach was also used in the case of modeling of profiled sheeting at researching of slabs.

Fig.1. Three-dimensional geometrical modelling of profiled sheeting H75J

Additionally for slabs ПМА-1, ПМА-3, ПМА-4 і ПМА-5 the geometrical model of profiled sheeting in a view of 3D body was constructed. For modelling of curvilinear reinforcement (ПМА-2) the standard instrument of Allplan "Bar shape" from module "Engineering" are used.

In addition, these slabs were modelled in LIRA software by another way, having constructed a line of profile and "pulling" it in the connected between planes and surfaces. The model constructed as a 0.7 mm thick plane, which corresponds to the profile of a profiled sheeting, which was then "stretched" to a length of 1500 mm by means of LIRA.

Further transformations of basic geometric models were associated with the modelling of the structure of concrete slabs. They consisted in breaking up the basic geometric models on the pentahedron and hexagonal shapes according to the shape and location of the reinforcement with LIRA tools. Such a breakdown performed by cutting the bodies with longitudinal planes (vertical, horizontal, and sloping). For modeling of curvilinear reinforcement, (ПМА-2) base bodies cut with curvilinear transverse surfaces.

As a result, they received hundreds of simple body shapes, on which superimposed the grid of the FE in LIRA software. On fig. 2 the geometric model of the ПМА-2 slab is given, completely ready to overlay the FE grids. On figure also, the section divided into simple planes and the total number of bodies of the model indicated.

Modeling of materials and contact was based on the results of experimental determination of physical and mechanical characteristics and on the behavior of materials during loading.

Amount of bodies – 6678

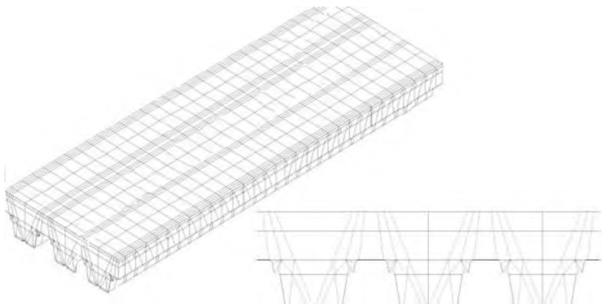

Fig.2. Geometrical model of slab ПМА-2

Bundle at modelling MFE for each material, it's necessary to set a model for it. The choice of a particular model of work depends on the type of calculation that is intended to be used for the research of the SSS in LIRA, and on the expected conditions of work and behavior of the material. The most common models are linear-static, nonlinear-elastic, elastic-plastic (bilinear) and plastic. In our case, for the study of all structures, both linear and non-linear static calculations were used. According to the last, part or all of the materials should be given as nonlinear indicating a model of nonlinear behavior.

Accepted for the calculation of the MFE material deformation diagrams were given in the LIRA as functions s - e, and then introduced as additional characteristics in the modulus of elasticity of materials.

The material of profiled sheeting and reinforcement– steel, that has the modulus of elasticity E=2,1·105 MPa and coefficient of Poisson ν=0,3. The shear modulus G is not set (it is calculated automatically). The material of the profiled sheeting is given as an isotropic non-linear, elastic-plastic, because the steel works equally on compression and tension. Such a model of the work of the material involves a two-line diagram of deformation, formed by a sloping elastic site and a horizontal plot corresponding to the yield line (Fig. 3). The boundary of yield f_y = 330 MPa was found experimentally. The boundary of strength in calculations is not used.

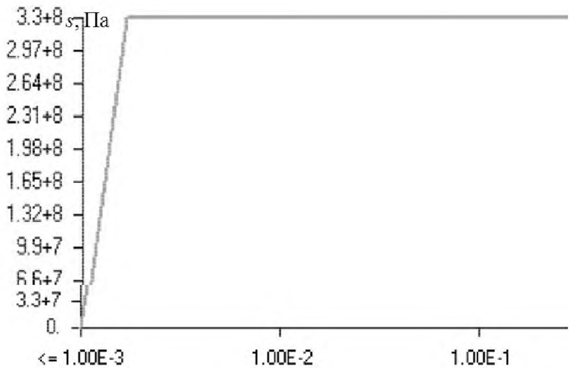

Fig. 3. Diagram s - e elastic-plastic model of profiled sheeting

A part of an array of concrete with a thickness of 0.3 mm used for modelling the contact. At the same time, to the boundary of the contact strength, its work corresponded to the work of concrete with the corresponding physical and mechanical characteristics, and after - the contact deformation was nowhere restricted, which corresponded to the segregation of materials and their independent deformation. The contact strength boundary was also determined experimentally and equaled 20 kPa. For contact, a nonlinear-elastic model of material work is used.

Table 1

Types of finite elements used to model the components of the slabs, and their properties

№	Component of slab	Type of FE	Property
1	Profiled sheeting	3D Solid	1) $E = 2{,}1 \cdot 10^5 MPa;\ \nu = 0{,}3$ * 2) $f_y = 330MPa$ 3) Stressed elastic model (fig.3) 4) Amount of nodes (five or six)
		2D Plate	1) $E = 2{,}1 \cdot 10^5 MPa;\ \nu = 0{,}3$ * 2) $f_k = 20MPa$ 3) Stressed elastic model (fig.3) 4) Thickness $t = 0{,}7mm$ 5) Amount of nodes (three or four)
2	Reinforcement concrete	3D Solid	1) $E_{ck} = 480MPa;\ \nu = 0{,}1$ * 2) $f_{ck} = 1MPa$ 3) Non-linear stressed model (fig.3) 4) Amount of nodes (five or six)
3	Contact	3D Solid	1) $E_k = 480MPa;\ \nu = 0{,}1$ * 2) $f_k = 20KPa$ 3) Non-linear stressed model (fig.3) 4) Amount of nodes (five or six)

4	Reinforcement	1D Beam	1) $E = 2{,}1 \cdot 10^5 MPa;\ \nu = 0{,}3$ * 2) $f_y = 330 MPa$ 3) Stressed elastic model (fig.3) 4) Shape and dimensions of transversal cross-section, its geometrical characteristics 5) Amount of nodes (two)
5	Load, support	Rigid	Absolute rigid FE, which doesn't change the shape, but only location in the space

**displacement modules were calculated automatically*

Plane (2D) FE – is an element that combines three or four adjacent nodes and is a triangle or quadrilateral respectively. Such FE are used for researching of SSS of plates and shells, i.e. bodies in which one size (thickness) is much smaller than the other two. Such elements allow determining the distribution of the parameters of the SSS on the surface (plane), but do not allow investigating the change in the stresses and deformations in thickness, replacing their values on the upper and lower surfaces of the FE. Pressure vessels, thin-walled structures, etc. can be examples of linear FE use. The main properties of plane FE include physical-mechanical characteristics of the material, the thickness of the plate or shell.

Volume (3D) FE – is an element that most often combines four, five, or six nodes. Such FE are used when it's necessary to research the SSS in each point of body. Often, they are used for research of massive three-dimensional bodies, in which the three dimensions are approximately the same. Such elements can be in the form of tetrahedral, hexahedral and prism. The physical-mechanical characteristics main properties of volume FE refer to the physical-mechanical characteristics of the material.

Absolute rigid FE (ARFE) – is an element that does not change its form under the action of the load (its form remains stable in all conditions), but it can change its relative position in space. ARFE necessarily consists of one related node and one or more driven. At displacement related node all AFRE moves without deformation in its direction. This property allows modeling the connection of model elements among themselves, but is most often used for modeling loads and supports.

Conclusions. Based on the performed calculations, the following practical recommendations for the model of SSS of CSRS with profiled sheeting of MFE have been developed: To use in calculations of FE in the form of hexaheders and prisms for which to develop the geometry into pentahedron or hexahedron. For modeling the curves to use the cross-sections when intersecting planes with curved surfaces, at modeling materials to consider their destruction in the form of incomplete displacement at a constant load (horizontal line), which corresponds to the boundary of strength or fluidity. To use nonlinear models of materials - elastic-plastic or non-linear-elastic, along with non-linear static calculations. Depending on the calculation conditions, use plane or volume FE for modeling the profiled sheeting. If it is necessary to investigate the initial moment of loss of stability and its shape, then 3D FE is used. In the case of 2D FE use for model of profiled sheeting work, the 2D FE grids applied to the corresponding planes and surfaces. To carry out an analysis of the results for individual components using interactive cross-sections.

List of literature:

1. Стороженко Л. І. Розрахунок сталезалізобетонних таврових балок з армуванням вертикальними листами за деформаційною схемою / Л. І. Стороженко, А. В. Іванюк, В. М. Чередніков, О. В. Череднікова // Вісник Одеської державної академії будівництва та архітектури.– Одеса:Зовнішрекламсервіс, 2012.– Вип. 47, частина 2. – С. 342 – 348.

2. Семко О. В. Чисельний аналіз напружено-деформованого стану тонкостінних сталевих елементів сталезалізобетонних конструкцій / О. В. Семко, Ю. О. Авраменко, О. В. Череднікова // Зб. наук. праць. Серія: Галузеве машинобудування, будівництво. – Полтава: ПолтНТУ, 2012. –Випуск 3 (33). – С. 212 – 217.

3. Срибних М. О., Некрасов О. В. Первый проект от эскиза до презентации / М. О. Срибних, О. В. Некрасов // Электронное учебное издание. — Екатеринбург: ООО Фирма «Уралкомплект – наука», 2014. –С. 210 – 220.

4. Сталезалізобетон. Continuance // Зб. наук. праць. – Полтава : ПолтНТУ, 2016. –358 с.

Анотація

В статті запропоновано чисельні методи розрахунку сталезалізобетонних плит з профільованим настилом, які являють собою неоднорідні композитні конструкції, для яких необхідно мати надійні та доведені до практичного застосування методи їх розрахунку, які б ураховували неоднорідну структуру елементів конструкцій, реальні схеми навантаження, крайові умови тощо.

Ключові слова: сталебетонні плити з профільованим настилом, неоднорідні композитні конструкції, чисельні методи розрахунку, напружено-деформований стан, метод скінченних елементів.

Аннотация

В статье предложены численные методы расчета сталежелезобетонных плит с профилированным настилом, которые представляют собой неоднородные композитные конструкции, для которых необходимо иметь надежные и доведенные до практического применения методы их расчета, которые бы учитывали неоднородную структуру элементов конструкций, реальные схемы нагрузки, краевые условия и т.п.

Ключевые слова: сталебетонные плиты с профилированным настилом, неоднородные композитные конструкции, численные методы расчета, напряженно-деформированное состояние, метод конечных элементов.

Стаття надійшла до редакції у лютому 2018 р.