https://doi.org/10.15407/scine17.03.028

DROBENKO, B. D. (https://orcid.org/0000-0001-7084-1196),
KUSHNIR, R. M. (https://orcid.org/0000-0003-2749-6846)
and MARCHUK, M. V. (https://orcid.org/0000-0001-8985-574X)
Pidstryhach Institute for Applied Problems of Mechanics and Mathematics,
National Academy of Sciences of Ukraine,
3-b, Naukova St., Lviv, 29060, Ukraine,
+380 32 263 8377, adm@iapmm.lviv.ua

APPROACH TO STUDYING THE STRENGTH AND DETERMINING THE YIELD STRESS OF ROCKET AND SPACE ENGINEERING STRUCTURES

Introduction. *Designing rocket structures requires computer modeling of their mechanical behavior in operating conditions. Based on the optimal design drawings obtained from computational experiments, a physical prototype has been made and tested. Depending on how successful the prototype passes the tests, the serial production of such structures is launched. The share of computer modeling in this process is constantly growing, since experimental studies are quite limited and extremely expensives.*

Problem Statement. *Estimates of structural strength significantly depend on the accuracy and reliability of data on their stress-strain state under operating conditions. Therefore, the development of software for assessing the stress-strain state of structures based on adequate mathematical models is extremely relevant.*

Purpose. *The purpose is to develop a method for studying the strength of complex structures of rocketry under intense loads and to determine ultimate breaking loads according to the results of computer modeling.*

Materials and Methods. *The problem is formulated within the framework of the geometrically nonlinear theory of thermoelastic plasticity, assuming that displacements and strains are large and stresses exceed the ultimate breaking load of materials. To solve the formulated problem, the finite element method has been used.*

Results. *A method for studying the stress-strain state of rocket complex structures under intense loads has been developed to estimate the ultimate breaking loads of such structures according to the results of computer modeling based on high-precision mathematical models. It has been successfully tested at the Yangel Pivdenne Design Office while designing fuel tanks of a launch vehicle.*

Conclusions. *The developed method makes it possible to significantly reduce or to completely abandon experiments during which the structure is carried to failure.*

Keywords: mathematical and computer simulation, space rocket technology, strength, and failure.

Designing modern structures is associated with a complex multi-stage process of mathematical and computer simulation of their mechanical behavior under operating condi-

Citation: Drobenko, B. D., Kushnir, R. M., and Marchuk, M. V. Approach to Studying the Strength and Determining the Yield Stress of Rocket and Space Engineering Structures . *Sci. innov.* 2021. V. 17, no. 3. P. 28—36. https://doi.org/10.15407/scine17.03.028

tions. In cyberspace, engineers and researchers practically look for a rational or optimal design of structure, study its mechanical behavior under different load conditions, materials, and parameters of its geometric shape by computer modeling. Based on the drawings of the optimal design obtained as a result of computational experiments, a physical prototype is made and subjected to comprehensive trials. As a result of successful tests, serial production is launched. The share of mathematical and computer simulation in this process is constantly growing, as experimental studies of the mechanical behavior of complex structures are quite limited and costly.

Estimates of the service life of structures significantly depend on the accuracy and reliability of data on their stress-strain state under operating conditions. Therefore, their mechanical behavior under operating conditions is studied with the use of refined mathematical models that, in addition to the conventional properties of elasticity of materials, take into account their plastic properties. This problem is especially relevant for the design of structural elements of rocket and space technology, in which contradictions between the requirements for strength and minimum material consumption are most striking. Designing structural elements, when the task is to determine the ultimate breaking load, at which the material of a significant part of the mentioned objects is in an elastic-plastic state, with strains and displacements reaching significant values, may be an example. This state of the material, which does not cause any failure of the functional purpose of the structure and the operation of its equipment, is allowed in some structural elements of disposable rocket and space technology and in the case of long-term operational load.

The study of structures given plastic strains and large displacements and strains allows the use of additional material resources, which makes it possible to increase the operational load in the design estimates.

Obtaining reliable results for the strength of real structures of rocket and rocket-and-space technology is associated with difficulties caused primarily by their subtlety and the presence of different types of geometric concentrators in the form of frames, stringers, and so on. In particular, the reinforcing elements essentially complicate the application of shell theories, because the stress-strain state in their vicinity during loading is significantly three-dimensional. The use in these areas of the assumptions of the theory of shells may lead to the difference of the obtained solutions from the actual stresses and strains. On the other hand, under intense loads, thin shells are strongly displaced as a rigid whole. Most finite elements of the general type react to such a displacement by the appearance of false shear and membrane strains, which often leads to a slowdown in convergence and "locking" of solutions [1]. The situation worsens as ratio of the finite element length to its thickness increases [2].

Many aspects of the behavior of complex mechanical structures are associated with the interaction of their various components. Often, they cannot be predicted or traced experimentally (either in computational or in field experiments) with individual separated elements [3]. As a rule, actual stresses in real structures significantly differ from those predicted by partial experiments with individual structural elements (because of structural continuity and availability of alternative load paths for individual elements).

The researchers of Pidstryhach Institute for Applied Problems of Mechanics and Mathematics (IAPMM) of the NAS of Ukraine have developed a method for studying the stress-strain state of rocket and space technology structures under the action of intense force load and for determining the ultimate breaking loads and zones from which they may start breaking up, based on the results of computer simulation within the general model of a geometrically nonlinear elastic-plastic body [3—8], with the use of the finite element method. The obtained results have been tested in operating conditions, in the course of tests of large fuel tanks and introduced into production at Yangel's *Pivdenne* Design Office.

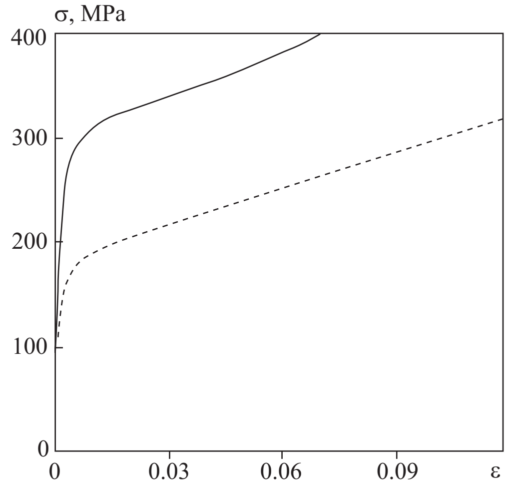


Fig. 1. Strain curves of materials of the cylindrical part of the tank (solid line) and bottoms (dash line)

The method for studying the strength of the structure is as follows. The study begins with simple 2D models. With the use of the documentation, a finite element model of the structure is built for chosen dimensions of its components and a model is designed from the bottom upwards, i.e. from points, lines to areas.

In numerical modeling of deformation processes, we use, as a rule, finite elements with a quadratic approximation of displacements [6]. While building a discrete model, we check to be certain that there are no strongly distorted or elongated finite elements (with a height-to-width aspect ratio that does not exceed 1: 2).

Experimental strain curves of the structure materials are used as input data as well. The strain curves are set pointwise, as interpolation splines [7, 8].

The conditions of fixing the structure are simulated and the load steps for the computational experiment are set.

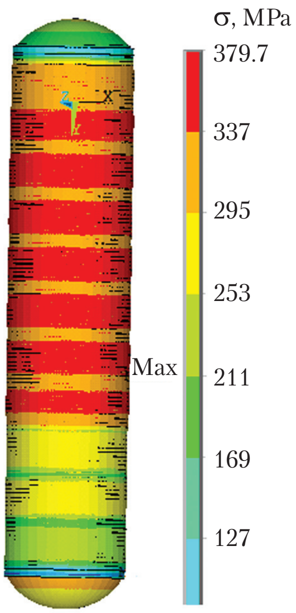
After the successful computational experiment, we have analyzed the obtained values of displacements, strains, and stresses in the nodes, determined the critical areas of the structure with the highest stresses, strains, and strength criteria as well as the places from which failure is likely to start (where the corresponding strength criteria reach maximum).

Having analyzed the results, we build a 3D fragment of the structure in the area with maximum values of parameters (stresses, strains, and criteria), for which we repeat the operations done for the 2D models (build a 3D finite element model, starting from points, lines, surfaces to volumes, set the conditions for fixing the fragment and the load). Having completed the computational experiment, we proceed to the analysis of the stress-strain state of a 3D fragment of the structure (determined the critical local areas of the structure with the largest deformations, stresses, strength criteria).

After a series of computational experiments, having determined the ultimate breaking load and the most stressful areas of the structure by computer simulation, we may perform a field experiment on a physical prototype for loads that are significantly less than the ultimate breaking ones. While performing these experiments, to measure deformations while applying loads, strain gages shall be placed in the most stressful places of the structure, as determined by computer simulation. After this, the experimental values of strains and stresses in these places shall be compared with similar values of the computational experiment. If the results of computational and field experiments coincide for loads less than the ultimate breaking ones, there is no need to increase the load to the ultimate breaking one in the physical prototype of the structure. This allows significantly reducing the costs of studying the strength problems of the elements of rocket and space technology.

To illustrate the proposed method, let us consider a large rocket structure, an eighteen-meter fuel tank of a space launch vehicle under the action of internal pressure.

The fuel tank is designed as a 3.9 m diameter cylindrical body closed by the upper and the lower spherical bottoms, each having a radius of 2.5 m. The body consists of eleven cylindrical shells of different types, with intermediate thickened zones between them for welding. As a result of the presence of circular frames and longitudinal strin-



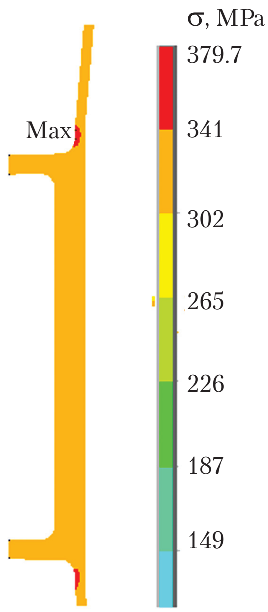


Fig. 2. Distribution of equivalent stresses in the tank as a whole body at a pressure of 0.84 MPa

Fig. 3. Equivalent stresses in the vicinity of the most stressed area

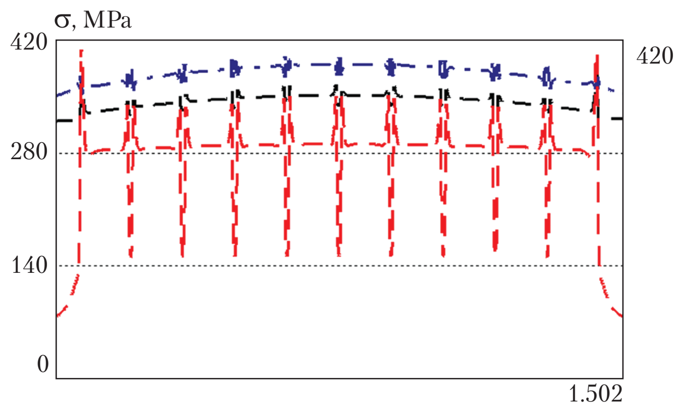


Fig. 4. Distribution of equivalent (solid line), axial (dash line) and circular (dash-and-dot line) on the inner surface of the fifth shell

gers, the inner surface of the shells has a wafer structure. The strain curves for the materials of which the wafer cylindrical shells and bottoms are made are shown in Fig. 1.

In order to quickly determine the strength and ultimate breaking load in the lower part, the whole tank deformation processes have been simulated based on the axisymmetric model, with the stringers neglected. For this purpose, we have used eight-node biquadratic finite elements [2].

Several computational experiments have been performed for different values of internal pressure. Fig. 2 shows the intensity of equivalent von Mises stresses in the tank at an internal pressure of 0.84 MPa. These stresses reach their absolute maximum (379.7 MPa) at the front of the rounded coupling of the first (immediately after the transition zone from the adjacent shell) frame with the inner surface (see Fig. 3) of the fifth shell (with a base thickness of 3 mm).

The stress distribution on the inner surface of the fifth shell is generally shown in Fig. 4. As one can see, there are local surges in the vicinity of each frame. The maximum stresses in the shell are mainly the circular ones that are highest in the middle of the shell. The axial stresses are generally lower, but in the vicinity of the extreme frames bordering the transitional thickened parts for welding individual shells, one can see their significant concentration. In the vicinity of these areas, they exceed the maximum circular stresses. In these places, equivalent von Mises stresses reach their maximum (at the level of the ultimate strength at a pressure of 0.84 MPa).

It should be noted that the absolute maximum stress in the fifth shell is greater than other local maxima in similar places of other shells of the same type by values of the order of computational error (maximum stresses in other shells range from 378.2 to 379.7 MPa). Practically, almost all these maxima can be considered equivalent, with the general trend being as follows: the local stress maxima are located in the vicinity of the extreme frames adjacent to the transition areas for welding individual shells. The stresses in the three lower shells (4.4 mm thick) and in the bottoms are significantly lower. The equivalent von Mises strain at the places of maximum stresses is 5.987%.

It should be noted that with a thickness of the upper bottom of 2.8 mm, the total displacement of the top part of this bottom reaches a value that is 73 times greater than the thickness of the bottom. The stresses at this point also exceed the ultimate strength for the bottom material (320 MPa, see Fig. 1). Increasing the thickness of the lower bottom by 0.2 mm eliminates this problem (at a thickness of 3 mm, the equivalent stresses in the upper bottom are less than the ultimate strength). The maximum deflections of the cylindrical shells in the central part of the tank reach twenty minimum thicknesses of these shells.

Therefore, having analyzed the results, we conclude that within the model of geometrically nonlinear, elastically and plastically deformable axisymmetric body (excluding longitudinal stringers) at a pressure of 0.84 MPa the equivalent von Mises stresses exceed the ultimate strength in the vicinity of the extreme frames of 3 mm thick shells. The axial stresses being highest at these local maxima, the tank breaks up in the circular direction (unlike in the case of conventional axial propagation of cracks in cylindrical shell-type bodies, when fracture is caused by circular stresses).

The results of the computational experiment based on the axisymmetric model have also shown that the action of the spherical bottom on the cylindrical part of the tank may be replaced by appropriately setting axial stresses at the end of the cylindrical component, distributing the whole load on the bottom at the ends of the transition area of the cylindrical part, i.e. by setting the axial stresses $\sigma_z = p\,(R - h\,)^2/(R^2 - (R - h)^2)$ at the end, where R is the outer radius of the cylindrical part of the tank and h is the thickness of the transition area.

It should be pointed out that the axisymmetric finite-element model has been analyzed for convergence. The calculations for different sizes of finite elements have been done. For 0.5 mm and 1 mm elements, the differences in the maximum equivalent von Mises stresses are less than 0.25%. This difference is also within 1% for 1 mm and 2 mm elements.

To study the effect of longitudinal stringers on the stress state of the tank, a similar study has been performed, with the effect of frames neglected. For this purpose, the tank's cross section in the middle of the area between the frames has been analyzed. From the symmetry conditions, half of the longitudinal stringer and half of the waffle cell in the circular direction have been considered (within the framework of plane strain problem). Local stress concentrations in the vicinity of the stringer have been found significantly lower than in approaches to the frames.

Given the current capabilities of computers, on the one hand, and the complexity of geometrically and physically nonlinear problems in 3D formulation, on the other hand, the tank strength in the vicinity of structural components in which maximum stresses occur has been studied on partial models in the axisymmetric formulation, and the comparative analysis of the obtained solutions with similar ones for the whole structure has been made. In particular, the stress-strain state of a shell fragment of five wafer cells has been studied. The action of the bottoms is replaced by the axial force at one end; the conditions of equality of displacements in the axial direction to zero are set on the opposite edge of the fragment.

Fig. 5 shows the general distribution of equivalent von Mises stresses in such a fragment of the shell at a pressure of 0.84 MPa. As you can see, in this formulation, we reach the same level of maximum internal pressure as while considering the whole tank of eleven shells. The maximum stress equivalents are reported in the vicinity of the rounded coupling of the first (after the thickened part) and the last (before the thickened part) frames of the considered fragment, as a result of surging axial stresses in these areas.

Therefore, based on the obtained results, we conclude that for the considered fragment of the five cells we reach the ultimate strength at a pressure of 0.84 MPa. Moreover, the obtained results are consistent with those obtained for the whole tank of 11 shells with an accuracy of 0.7%. Similar results have been obtained for structures consisting of one and two complete shells (10 waffle cells in each).

To summarize the obtained results, the stress state of the same fragments of the shell of five cells has been calculated based on the model of a 3D

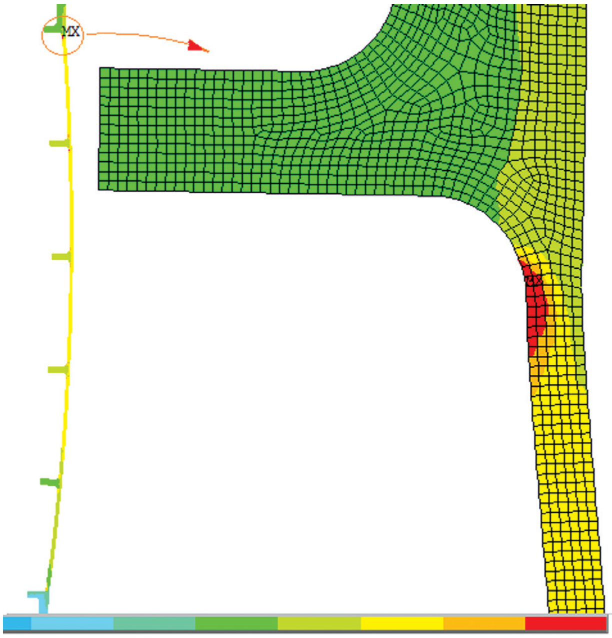


Fig. 5. Equivalent stresses in the five-cell shell fragment

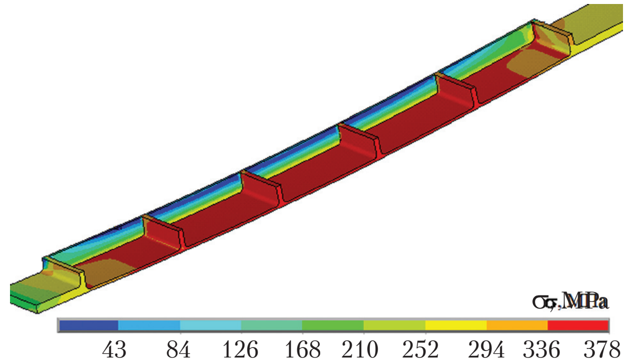


Fig. 6. Equivalent stresses in 3D shell fragment at a pressure of 0.88 MPa

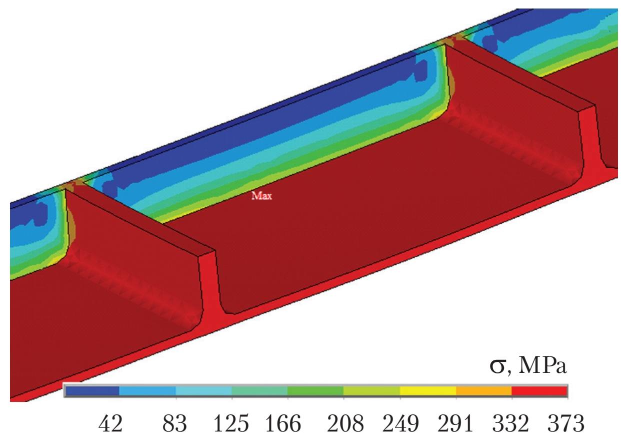


Fig. 7. Equivalent stresses in the median wafer cell of the shell at a pressure of 0.88 MPa

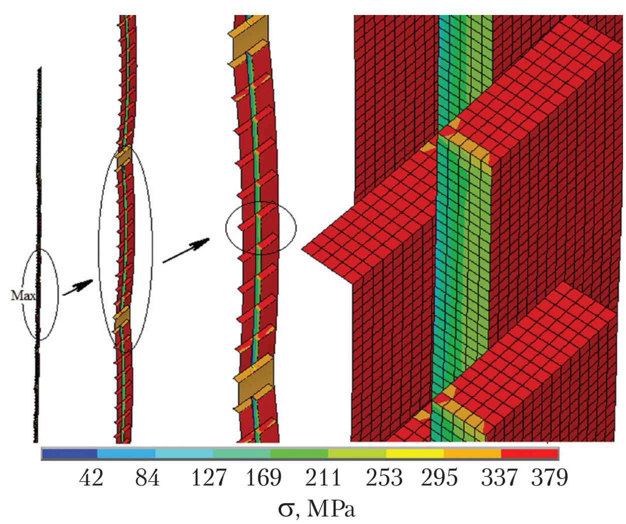


Fig. 8. Equivalent stresses in the cylindrical body at a pressure of 0.87 MPa in the framework of the shell model

geometrically nonlinear elastic-plastic body. From the circular symmetry conditions, the sector corresponding to half of the wafer cell (dimensions 0.137 m x 0.136 m) has been considered (see Fig. 6). The *Y* axis is directed along the axis of the cylindrical part of the tank; the *X* axis is oriented along the thickness; the Z axis coincides with the cylindrical angular coordinate at $z = 0$. Axial stresses that take into account the influence of the spherical bottom and zero axial displacements are set at the lower edge of the cylindrical part and at the opposite edge, respectively.

The nature of the stresses on the inner surface in the middle between the longitudinal stringers (in the plane of circular symmetry) is similar to that of those previously obtained within the axisymmetric model. The maximum stresses are the circular ones. In the vicinity of the extreme frames, in front of the transition area, there recorded a surge of the axial stresses. However, while studying the 3D stress state of the tank fragment under pressure, it has been established that the equivalent Mises stresses reach their absolute maximum in other places, namely, in the middle of the studied fragment, on the inner surface of the central cell, 5 mm from the stringer (see Fig. 7). The same qualitative result has been obtained for a cylindrical fragment of the tank which consists of two whole shells, under similar load conditions.

Having made calculations for different values of internal pressure within the 3D model representation, the pressure (0.88 MPa) at which the maximum equivalent von Mises stresses reach the ultimate strength is determined. The highest value of the maximum stress criterion for the critical load is 0.985, which also indicates approaching failure according to this strength criterion.

It should be noted that as we approach the ultimate breaking load, the stresses over the tank thickness in the areas of maximum stresses even up, and the stress gradient through thickness practically disappears, as the difference between equivalent stresses on the inner and the outer surfaces is about 1—2 MPa, while in the area of elastic strain, it exceeds 20 MPa.

The influence of variation of geometrical parameters of shells, frames, and stringers on the stress state of the tank in the framework of the 3D model has been also studied. For example, the thickness of the thinnest shell should range within 3—3.35 mm; the thickness of frames and stringers should not exceed 6.4—6.9 mm. The computational experiment has been performed for the case where the thickness of shells, frames, stringers, and transition areas is taken at the upper limit of their allowable variation. Pressure at which the equivalent stresses approach the ultimate strength under such conditions is equal to 0.92 MPa.

Similarly, the tank strength has been studied within the framework of the six-modal theory of shells [9]. Based on the cyclic symmetry conditions, one cell of the shell is considered in terms of the angular coordinate. The surface of the shell, parts of the frames and stringers are represented by shell elements (in each node of the finite element division of the considered area, we have three displacements, two angles of rotation of the normal and its compression). The impact of the bottom on the cylindrical body is replaced by the axial stresses set at the upper edge of the body; at the other edge zero axial displacements are set. The conditions of cyclic symmetry are given at the edges as $\alpha = 0$ and $\alpha = Room2 / R$, where *Room*2 is the linear size of the cell in the circular direction, R is the tank radius. The pressure is set on the inner surface of the cylindrical body.

The computational experiments have shown that the equivalent stresses are almost close to the ultimate strength at a pressure of 0.87 MPa (see Fig. 8). As one can see, the maximum stresses are recorded in the middle of the fifth shell and they are the circular ones.

It should be pointed out that finite-element solutions in the case of the shell model are quite stable in terms of the size of finite elements. The same value of ultimate breaking load has been obtained for maximum finite element size of 2 mm and 5 mm.

The developed method and the corresponding software have been tested and implemented in the Yangel *Pivdenne* Design Office. With their help, the stress-strain state of fuel tanks has been studied. The comparative analysis of the results of computer simulation and field experiments has shown a good agreement in terms of the location and the nature of failure, as well as the value of ultimate breaking pressure. The last measured value of pressure before the failure of physical prototype is 0.89 MPa (in the computational experiment based on the 3D model a critical load of 0.88—0.92 MPa, depending on the tolerance for the shell thickness, has been obtained). On practice, the tank prototype starts failure with the fifth shell, as predicted by the computer simulations.

Hence, a method for studying the strength of structures and for determining ultimate breaking load by computer simulation and non-destructive experiments has been developed. Assuming that displacements and strains may be large and stresses significantly exceed the the yield stress of materials, the problem has been formulated within the geometrically nonlinear theory of thermoelastic plasticity. The finite element method has been used to solve the problem stated. Appropriate software has been developed on this basis.

Within the framework of the developed method, the stress-strain state of a fuel tank of launch vehicle under the action of internal pressure has been studied for different model assumptions: the tank has been considered an axisymmetric structure, a composed shell, and a 3D elastic-plastic body. Quantitative estimates of its strength have been obtained; the ultimate breaking load and the most loaded local areas have been determined.

The analysis of the results of computer simulation of the fuel tank deformation processes allows us to state that the maximum stresses at ultimate breaking loads are located in the cylindrical part of the tank at the level of the fifth shell; in the vicinity of the spherical bottoms and spacer frames to which the cylindrical part of the tank is attached, the stresses are less than in its cylindrical part. It is the fifth shell from which the tank starts breaking up as the load increases, which has been confirmed by experimental tests. The stresses in the lower wafer shells are significantly lower.

With the transition to the area of plastic strains, as the load increases, the difference between the stresses on the inner and the outer surfaces decreases, which may also indicate an approach to failure (in the early stages, the stress on the inner surface is significantly higher).

The proposed method makes it possible to sharply reduce, or even to abandon, the full-scale experiments, during which the structure is carried to failure. After computational experiments and determination of the ultimate breaking load and the most stressful places of the structure by computer simulation, one may perform a field experiment on a physical prototype for loads that are significantly less than the ultimate breaking one. In these experiments, the sensors shall be placed in the most stressful places of the structure, as determined by computer simulation. The experimental values of strains and stresses in these places shall be compared with similar values of the computational experiment. In the case of coincidence of the results of computational and field experiments for the loads less than the ultimate breaking one, there is no need to increase the load to the ultimate breaking one in experiments with a physical prototype of the structure. This enables saving valuable materials and money.

The research has been carried out within R&D project No. 1 *Development of an Approach to Assessing the Mechanical Condition and Determining the Actual Ultimate Breaking Loads of Shell Structures Based on Computer Simulations and Non-Destructive Experiment Tests* of the NAS of Ukraine dated April 1, 2019.

REFERENCES

1. Bathe, K. J. (1995). *Finite Element Procedures Analysis*. Englewood Cliffs: Prentice Hall. 1037 p.
2. Zienkiewicz, O. C., Taylor, R. L. (2000). *Finite Element Method*: *V. 1. The Basis*. London: Butterworth Heinemann. 689 p.
3. Buryk, O. O., Drobenko, B. D. (2016). Stress-strain state of the elements of building structures in the case of fire. *Journal of Mathematical Science,* 217(3), 330—344. doi: 10.1007/s10958-016-2976-x.
4. Budz, S. F., Drobenko, B. D., Mychailyshyn, W. S. (1992). Computer simulation of mechanical system thermoelastic-plastic behavior. *Preprint 34—89*. IAPMM AS USSR. 60 p. [in Russian].
5. Gachkevich, O., Drobenko, B., Kazaryan, K. (2003). Mathematical simulation of thermomechanical processes in conducting axially symmetric bodies under electromagnetic loadings. *Mechanical Engineering*, 4, 3—7 [in Ukrainian].
6. Hachkevych, O., Drobenko, B. (2010). *Thermomechanics of magnetizable electrically conductive thermosensitive solid*. V. 4. (Eds. Ya. Yo. Burak, R. M. Rushnir). Lviv: SPOLOM. 256 p. [in Ukrainian].
7. Drobenko, B., Hachkevych, O. (2014). Thermomechanics of electroconductive solids. In *Encyclopedia of thermal stresses* (Ed. Richard B. Hetnarsky). V. 11, Springer: New York, London. P. 6052—6063. doi: 10.1007/978-94-007-2739-7.
8. Drobenko, B., Vankevych, P., Ryzhov, Y., Yakovlev, M. (2017). Rational approaches to high temperature induction heating. *International journal of engineering science*, 117, 34—50. doi: 10.1016/j.ijengsci.2017.05.001.
9. Grygorenko, Ya. M., Vasylenko, A. T. (1981). *Theory of shells of variable stiffness*. Kyiv. 544 p. [in Russian].

Received 22.04.2020
Revised 25.08.2020
Accepted 23.02.2021

Б.Д. Дробенко (https://orcid.org/0000-0001-7084-1196),
Р.М. Кушнір (https://orcid.org/0000-0003-2749-6846),
М.В. Марчук (https://orcid.org/0000-0001-8985-574X)
Інститут прикладних проблем механіки і математики ім. Я.С. Підстригача НАН України,
вул. Наукова, 3 б, Львів, 29060, Україна,
+380 32 263 8377, adm@iapmm.lviv.ua

МЕТОДОЛОГІЯ ДОСЛІДЖЕННЯ МІЦНОСТІ ТА ВИЗНАЧЕННЯ РУЙНІВНОГО НАВАНТАЖЕННЯ КОНСТРУКЦІЙ РАКЕТНО-КОСМІЧНОЇ ТЕХНІКИ

Вступ. Проєктування ракетних конструкцій передбачає процес комп'ютерного моделювання їхньої механічної поведінки за умов експлуатації. За кресленнями оптимального проєкту, отриманого в результаті обчислювальних експериментів, виготовляють фізичний прототип, який піддають випробуванням, за результатами успішності яких переходять до виготовлення серійної продукції. Питома вага комп'ютерного моделювання в цьому процесі постійно зростає, позаяк експериментальні дослідження є доволі обмеженими і високовартісними.

Проблематика. Оцінки міцності конструкцій істотно залежать від точності й достовірності даних про їхній напружено-деформований стан за умов експлуатації. Тому розроблення програмного забезпечення для оцінювання напружено-деформованого стану конструкцій на основі високоточних математичних моделей є надзвичайно актуальним.

Мета. Розроблення методології адекватного дослідження міцності складних конструкцій ракетної техніки за інтенсивних силових навантажень та визначення руйнівних навантажень за результатами комп'ютерного моделювання.

Матеріали й методи. За припущення, що переміщення й деформації є великими, а напруження перевищують межу пластичності матеріалів, задачу сформульовано в межах геометрично нелінійної теорії термопружно-пластичності. Для її розв'язування використано метод скінченних елементів.

Результати. Розроблено методологію дослідження напруженого стану складних конструкцій ракетної техніки за інтенсивних силових навантажень з метою оцінювання руйнівних навантажень таких конструкцій за результатами комп'ютерного моделювання на основі уточнених математичних моделей, яку успішно апробована на Державному підприємстві «Конструкторське бюро «Південне» ім. М.К. Янгеля» при проєктуванні паливних баків ракети-носія.

Висновки. Розроблена методологія дає можливість суттєво скоротити або й взагалі відмовитись від натурних експериментів, під час яких конструкцію доводять до руйнування.

Ключові слова: математичне й комп'ютерне моделювання, ракетно-космічна техніка, міцність, руйнування.