




DOI: 10.22363/1815-5235-2023-19-2-220-232
 EDN: CTXRYX
 UDC 519.6:69:624.074

RESEARCH ARTICLE / НАУЧНАЯ СТАТЬЯ

Modern software features for shape optimization of shells

Evgenia V. Ermakova  , Marina I. Rynkovskaya 

RUDN University, Moscow, Russian Federation

 ermakova-ev@rudn.ru

Article history

Received: January 29, 2023

Revised: March 21, 2023

Accepted: April 5, 2023

Abstract. Shape optimization, as one of the types of structural optimization problems, is an important process in the design of shells, since it contributes to the creation of a structure with fine performance characteristics, expansion of design variations and knowledge base to obtain high-quality results. To solve the problems associated with determining the shape and creating more advanced structures, software packages include a special optimization module, which can be based on one or more mathematical methods, the purpose of which is to provide the best solution in the shortest possible time. The research is focused on the process of shape optimization in three well-known universal software packages: Ansys Mechanical, COMSOL Multiphysics and Simulia Abaqus, as well as in Rhinoceros modeling software with a special visual Grasshopper plugin. The purpose of the study is to analyze the technology of shape optimization in four software packages and to compare them with each other in terms of the problem-solving process, user interface, the fullness of libraries, accessibility for educational purposes and system requirements for a computer. The authors specify and describe the characteristic features of each software package. It was found that all the software packages under consideration are equipped with great opportunities for shape optimization of structures and have a variety of functionality for solving this type of tasks. The development of optimization technology in calculation and modeling software packages will allow obtaining the most effective solutions in the process of designing shells of complex shapes.

Keywords: structural optimization, optimal shape, optimization module, optimization methods, software packages, calculation, modeling


For citation

Ermakova E.V., Rynkovskaya M.I. Modern software capabilities for shape optimization of shells. *Structural Mechanics of Engineering Constructions and Buildings*. 2023;19(2): 220–232. <http://doi.org/10.22363/1815-5235-2023-19-2-220-232>

Современные возможности программного обеспечения для оптимизации формы оболочек

Е.В. Ермакова  , М.И. Рынковская 

Российский университет дружбы народов, Москва, Российская Федерация

 ermakova-ev@rudn.ru

История статьи

Поступила в редакцию: 29 января 2023 г.

Доработана: 21 марта 2023 г.

Принята к публикации: 5 апреля 2021 г.

Аннотация. Оптимизация формы как один из типов задач структурной оптимизации является важным процессом при проектировании оболочек, поскольку способствует созданию конструкции с хорошими эксплуатационными характеристиками, расширению вариантов дизайна и базы знаний

Evgenia V. Ermakova, Assistant, Department of Civil Engineering, Academy of Engineering, RUDN University, Moscow, Russian Federation; ORCID: 0000-0003-4515-6220; ermakova-ev@rudn.ru

Marina I. Rynkovskaya, PhD, Docent of the Department of Civil Engineering, Academy of Engineering, RUDN University, Moscow, Russian Federation; ORCID: 0000-0003-2206-2563; rynkovskaya-mi@rudn.ru

Ермакова Евгения Владимировна, ассистент, департамент строительства, инженерная академия, Российский университет дружбы народов, Москва, Российская Федерация; ORCID: 0000-0003-4515-6220; ermakova-ev@rudn.ru

Рынковская Марина Игоревна, кандидат технических наук, доцент, доцент департамента строительства, инженерная академия, Российский университет дружбы народов, Москва, Российская Федерация; ORCID: 0000-0003-2206-2563; rynkovskaya-mi@rudn.ru

© Ermakova E.V., Rynkovskaya M.I., 2023



This work is licensed under a Creative Commons Attribution 4.0 International License
<https://creativecommons.org/licenses/by-nc/4.0/legalcode>

для получения высококачественных результатов. Для решения проблем, связанных с определением формы и созданием более совершенных конструкций, в расчетные программы входит специальный оптимизационный модуль, который может основываться на одном или нескольких математических методах, цель которых обеспечить лучшее решение в кратчайшие сроки. Исследуется процесс проведения оптимизации формы в трех известных универсальных расчетных программах: Ansys Mechanical, COMSOL Multiphysics, Simulia Abaqus, а также в программе для моделирования Rhinoceros со специальным визуальным плагином Grasshopper. Анализируются технологии оптимизации формы в четырех программных комплексах, проводится их сравнение по процессу решения задачи, пользовательскому интерфейсу, наполненности библиотеками, доступности в учебных целях и системным требованиям к компьютеру. Выделяются и описываются характерные особенности каждой программы. Установлено, что все рассматриваемые программные комплексы снабжены большими возможностями для проведения оптимизации формы конструкций и имеют расширенный функционал для решения такого типа задач. Развитие технологии оптимизации в программах для расчета и моделирования позволит получить наиболее эффективные решения в процессе проектирования оболочек сложных форм.

Ключевые слова: структурная оптимизация, оптимальная форма, оптимизационный модуль, методы оптимизации, программы, расчет, моделирование

Для цитирования

Ermakova E.V., Rynekovskaya M.I. Modern software capabilities for shape optimization of shells // Строительная механика инженерных конструкций и сооружений. 2023. Т. 19. № 2. С. 220–232. <http://doi.org/10.22363/1815-5235-2023-19-2-220-232>

1. Introduction

Structural optimization is an iterative process of finding the best shape of an object during the design stage. Its dynamism has never been so intense as in recent decades, where the increase in the cost of raw materials and the necessity to decrease energy consumption have made it more decisive than ever to optimize the produced components starting from the early stages of design so that they fulfil their intended purpose for a minimum amount of constituent material [1]. The concept of optimization is based on mathematical principles and methods. A brief description of the main stages of optimization concept development is provided in [2; 3] and the most popular optimization algorithms used in various tasks related to civil engineering are contained in [4; 5]. The general optimization problem is to find the values of the control variables that minimize or maximize the objective function $f(x)$, taking into account some constraints. The goal of optimization is to create a new optimized design that will be the most efficient compared to the first one in terms of stress-strain state, strain energy, volume, etc. An overview of the studies is focused on structural optimization in civil engineering, which is shown in detail in [6], and the introduction to the process of structural optimization is given in [7]. There are three main types of optimization tasks based on design variables: size optimization, topology optimization and shape optimization, which are powerful tools for the effective construction design of buildings and structures (Figure 1).

Size optimization is a structural process in which the geometric parameters of the model under study are changed while maintaining the original appearance and topology. The properties of elements, materials and compounds of the object of study can be set as constructive variables. The integrity of the model itself remains unchanged. The problem of size optimization is formulated, for example, as the problem of determining the optimal distribution of the plate thickness or the rod's cross section [8].

Topology optimization finds the optimal distribution of material and voids in a specific area, considering the geometric space of the structure, loads and boundary conditions. This type of optimization is convenient when significant changes are expected in the model (the appearance of holes, merging elements, adding new components, etc.). Applying topology optimization to structural design typically involves considering characteristics such as weight, stresses, stiffness, displacements, buckling loads, and resonant frequencies, with some measure of these defining the objective function and others constraining the system [9]. Topology optimization is currently under active research. Among the articles that studied topology optimization, [10] focuses on new developments and improvements in the field of topology optimization and [11] discusses the issues of improving the applicability of topology optimization to structural design problems.

During the shape optimization process, the boundaries, shapes and contours of the object under study are changed while maintaining its geometric structure – topology and specified boundaries according to specific requirements. The article studies this type of problem since the model under consideration is a spatial covering consisting of a shell where holes cannot be formed. The identification of a maximally stable (or most lightweight, etc.)

structure is a natural task in shape optimization [12–13]. An introduction to this type of structural optimization is briefly presented in [14], and [15] reviews some fundamental ideas and recent developments in the field of shape optimization.

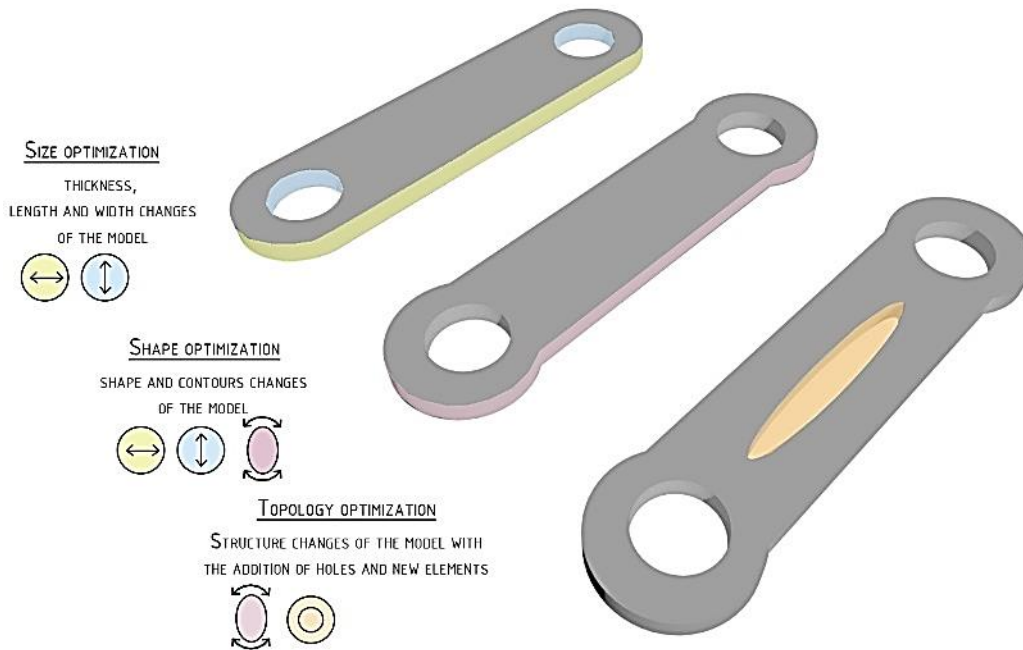


Figure 1. Three types of structural optimization problems

However, not so many practical works have been found that study particularly the process of shape optimization in the software packages COMSOL Multiphysics, Ansys Mechanical, Simulia Abaqus and Rhinoceros with Grasshopper. In [16] COMSOL Multiphysics is mentioned as part of the optimization solution for several examples of problems and applications on a specific topic, ranging from thermal conductivity equations to fluid mechanics problems. The calculation using COMSOL is also demonstrated in [17] where the process of optimization of the shape and geometry of a piezoelectric cantilever energy harvester is shown. Author of [18] research shape optimization of a free geometry surface in Ansys Mechanical using a program in the Fortran language, and in [19] the shape optimization of the model constructed according to the hyper equations is shown. In [20] shape optimization and topology optimization of the shell are carried out using the ATOM solver in Abaqus where the main attention is paid to the results of the experiments carried out, and the optimization process itself is almost not described. An example of using Abaqus with the Tosca Structure module for topology optimization of a part from the aerospace industry is shown in [21]. The optimization possibilities in Rhinoceros are studied in [22; 23], in which a general overview of existing Grasshopper plugins based on various methods is given. In [24; 25] the possibilities of linking Grasshopper with Galapagos for solving specific problems are considered.

It should also be noted that no works on the comparison of all the above-mentioned software packages for the implementation of the shape optimization process among themselves have been found in open access sources. However, there are similar studies, for example [26], dealing with the review of software packages, including Ansys, COMSOL Multiphysics and Abaqus to identify the most suitable software for modeling and calculation of composite materials. In [27] three software packages for structural optimization are evaluated: Genesis, MSC.Nastran and Optistruct.

Thus, **the purpose of the study** is to select and then compare well-known software packages for calculation and modeling, in which there is an opportunity to optimize the shape of the shell.

To do this, it is necessary to solve the following tasks:

- evaluate the overall implementation of shape optimization modules in software packages for calculation and modeling;
- show the functionality of the software packages and the main stages during the shape optimization;
- compare software packages with each other.

2. Methods

In order to create an optimal design option, the calculations are closely related to the design of the object under study. In the age of information technology, this whole process is performed in special software packages, the possibilities of working in which are constantly expanding. In most cases, software packages have versatility and involve solving a wide range of tasks in various technical fields. Software packages such as COMSOL Multiphysics 6.1, Ansys Mechanical 2022/R1 and Simulia Abaqus 2022 have a multifunctional system for automatization of the design process and allow solving various tasks, including structural optimization problems, as well as creating an initial version of geometry before calculation. However, not all software packages are currently equipped with the capabilities to carry out the optimization process. For example, LIRA-SAPR 2022 and SCAD Office 21.1, have no special module for optimization, despite their integrated environment for connecting calculation with design tasks and the ability to model shells by equations. APM FEM 20.0 software, which is a stress-strength analysis system for Compas 3D v21, includes an option for topology optimization. Other types of structural optimization are not available there yet.

As for software packages for 3D modeling, it is already possible to carry out optimization calculations directly in them. An example is Rhinoceros 7 with Grasshopper, which makes it possible to provide various types of optimization calculations, including shape optimization. Also, Rhinoceros 7 with Grasshopper has a connection with software packages for BIM design – ArchiCAD 25.0 and Revit 2023. The capabilities of modern software packages for shape optimization are presented in Table 1.

Table 1

The capability of shape optimization in software packages for calculation and modeling

Software package	Modeling by parametric equations	Shape optimization	Optimization method	Optimization module
SCAD Office 21.1	+	–	–	–
LIRA-SAPR 2022	+	–	–	–
APM FEM 20.0	–	–	–	–
Ansys Mechanical 2022/R1	+ (APDL)	+	Sequential Convex Programming-SCP, optimality criteria	Workbench-structural optimization
COMSOL Mutiphysics 6.1.	+	+	Method of Moving Asymptotes (MMA, GCMMA)	Shape optimization
			Sparse Nonlinear OPTimizer (SNOPT)	
			Interior Point OPTimizer (IPOPT)	
Rhinoceros 7 with Grasshopper	+	+	Evolutionary algorithm (EA)	Galapagos
Simulia Abaqus 2022	–	+	Condition-based optimization algorithm	SIMULIA Tosca Structure Shape

COMSOL Multiphysics 6.1 is a powerful universal software for solving simple and complex types of tasks, including interdisciplinary ones in all areas of engineering, industrial and scientific research. The software includes modules that extend the COMSOL modeling environment with customizable physical interfaces that solve problems in the field of structural mechanics and solid mechanics. It contains a specific interface for modeling shells, membranes, beams, plates, trusses, and pipes [28]. One of the key advantages of the COMSOL package is multiphysics – the ability to solve related interdisciplinary tasks in a single environment [29]. Even though the software package is quite well-known, it does not have a freely distributed version for educational purposes, but there is a demo version of the software provided on request.

COMSOL Multiphysics 6.1. has extensive optimization capabilities. The software package contains a multi-purpose extension – module Optimization, combined with all other modules of the software and supplementing the basic functionality. The module provides parametric, topology, geometric optimization and some types of inverse problems related to the choice of variable values based on external, usually experimental data.

Optimization calculations are performed in three solvers: Optimization, Shape Optimization and Topology optimization.

The Shape Optimization solver is responsible for the shape optimization process, which includes three gradient optimization methods used to solve problems with a large number of input parameters: Method of Moving Asymptotes (MMA, GCMMA), Interior Point OPTimizer (IPOPT), Sparse Nonlinear OPTimizer (SNOPT).

The general process of form optimization is shown in Figure 2, which can be divided into four main stages.

The first three steps are related to the initial geometry of the model, and the fourth step is already with the optimized version. It should be noted that moving from one stage to another in COMSOL is carried out by sequentially creating files in the Model Builder environment.

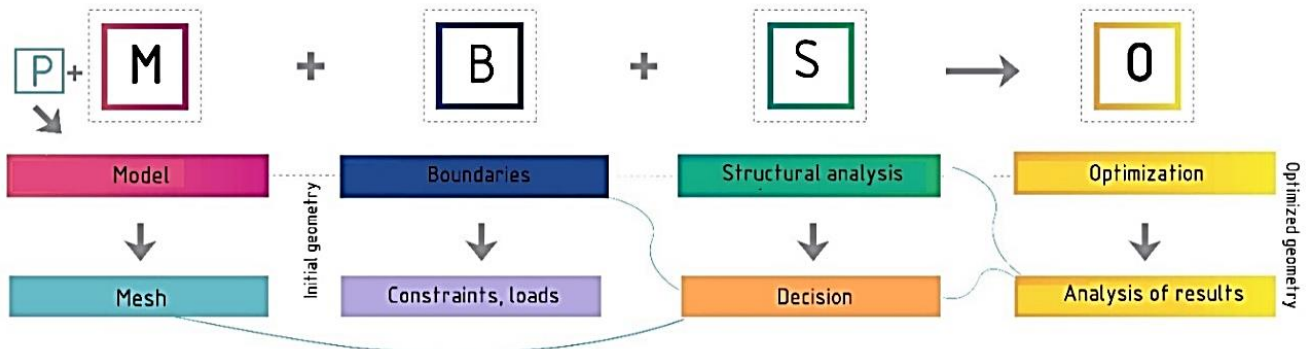


Figure 2. The process of shape optimization in COMSOL Multiphysics

All project data is collected in Model Builder, which allows to manage and add new functions using either analytical (in the form of partial differential equations) or numerical (in the form of matrices, graphs, algorithms, and optimization methods) model [30].

Before working with the object of study, the input parameters of the model are determined, which may include radius, loads, mass, etc., and they can then be set as criteria. After constructing the mesh, loads and boundary conditions are set in the model, and it is possible to proceed to structural analysis. After the calculation and analysis of the results obtained, a suitable optimization solver is selected – Shape Optimization and all other settings are defined inside this solver. To begin with, the appropriate method is chosen from the three methods (MMA, SNOPT and IPOPT), taking into account the type of task. For each method, it is necessary to determine the number of iterations consisting of repeated calculation steps to assess the compliance of the calculation results with certain quality criteria. The process continues until certain conditions are met. The most important and key point in the process of optimization of the mathematical model is the definition and selection of the objective function, which will be the criterion indicating the quality of the system and to which the project will be optimized. Next, the type of optimization task is selected: minimization or maximization; a system of restrictions is introduced. After determining all the data for the calculation, the optimization is launched and the subsequent analysis of the results is performed.

It should be noted that the practical application of the COMSOL program is described in [31], which shows the process of shape optimization of two shells: a sphere and a helicoid.

Ansys Mechanical 2022/R1 Ansys Mechanical 2022 / R1 is an advanced Finite Element Analysis (FEM) software package that provides solutions in various fields of engineering. The main advantages of the ANSYS software products are a high degree of integration of individual applications, an intuitive interface and support for high-performance computing [32]. In the construction industry, Ansys can be used to solve a wide range of tasks. For example, calculation of complex spatial above-ground structures of a building or solution of geotechnical problems, such as calculating the fastening of a pit, various foundations, slope stability with a structure, filtration calculations, etc. [33]. Unlike COMSOL, a software version of Ansys Student is distributed for educational purposes and provides free access to the Ansys package based on Ansys Workbench with basic applications.

The interface of Ansys, created based on the development of Motif Standard, provides convenient interactive access to the functions, commands, documentation and reference materials, a kind of guide is created that teaches how to use the software package step by step during the analysis [34].

In addition to carrying out calculations on structural strength, thermodynamics, mechanics of liquids, and gases, the software also implements structural optimization. There are several types of topology optimization available for solving problems, as well as shape optimization.

To calculate the shape optimization in the Ansys Workbench, two Static Structural and Structural optimization blocks are selected, which are connected, and four stages are sequentially performed through them to obtain an optimized model. The algorithm of steps is shown in Figure 3.

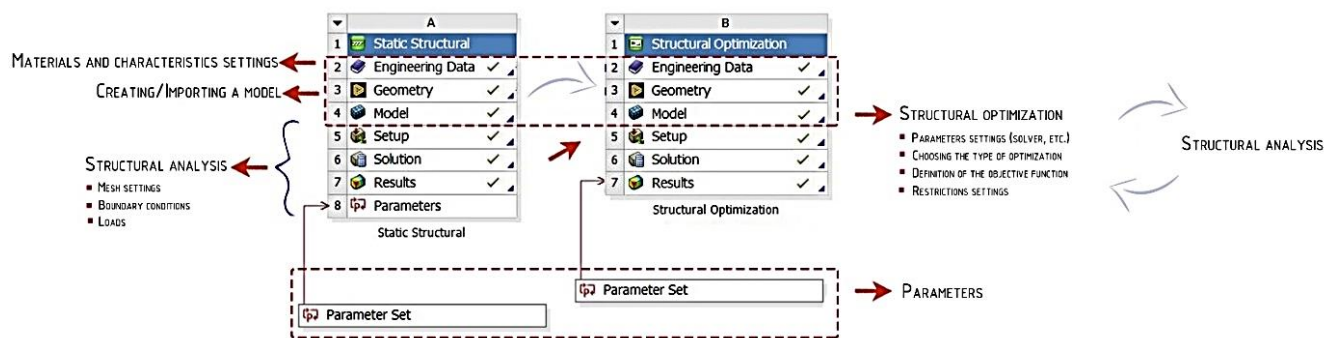


Figure 3. The process of shape optimization in Ansys Mechanical

Firstly, a model created in another software is imported into Static Structural, or the model is created from scratch in Ansys. The software package includes two tools for working with CAD geometry – Ansys Design-Modeler and Ansys SpaceClaim. Then, certain properties and materials of the model are set through the engineering data service Engineering Data. After installing the preparatory actions, the transition takes place to the Structural optimization block, where the necessary optimization method is first assigned. There are three types of methods available in Ansys for the iterative form search process: the sequential quadratic programming method, the optimality criterion method and the automatic solver, which is installed by the software. Next, the type of optimization, the number of iterations, and the time are set. After the criteria and constraints are determined (by weight, by displacement of the structure, etc.), the process of optimization is started. At the end, after processing in the postprocessor, the results are reviewed and, if they do not satisfy the task, then the reverse transition to model editing occurs.

Furthermore, additional software for CAD geometry Caeses was developed for Ansys, which has the capabilities for parametric modeling and includes tools for performing calculations and organizing the process of shape optimization.

Simulia Abaqus 2022. Simulia Abaqus Unified FEA is a multi-purpose software package that solves the problems of dynamics and strength by the finite element method. The versatility of Abaqus makes it possible to use it effectively at all stages of designing and creating modern products, as well as in almost all calculation, design and technological departments of enterprises [35]. This software is widely used in the construction industry, since it can be used to perform a detailed analysis of building structures under the influence of various types of loading, to obtain a reliable forecast of deformations of natural material and to conduct engineering studies of the dynamics of buildings and structures. Similar to Ansys, there is a special full-featured version of the Abaqus Student Edition software for educational purposes containing all the main modules and solvers but with a limit on the number of computational elements and nodes that should not be more than 1000.

The work in the software package is based on the modular principle, which makes working process in Abaqus more visual. Abaqus contains two main solvers – Abaqus/Standart and Abaqus/Explicit, as well as a powerful pre-postprocessor Abaqus/CAE. An analysis of the possibilities for topology optimization and optimization of the shape of the Abaqus package on the example of a simple part is given in [36].

Currently, the Simulia Abaqus 2022 version implements two types of optimization: parametric, which is available in a special Isight package, and nonparametric, which is implemented in the Tosca package. The Tosca Structure module is integrated into the Abaqus/CAE pre-postprocessor and has wide possibilities for topology, sizing, shape and bead optimization. One condition-based optimization algorithm is available in the system for shape optimization.

Abaqus uses Python scripts to provide interaction between modules such as Abaqus/CAE, Abaqus/Viewer, and various solvers. It is also possible to use Python scripts without logging into graphical interfaces (CAE and Viewer).

The general process of step-by-step implementation of shape optimization in Abaqus is shown in Figure 4. Before conducting the optimization process, the model goes through a preparatory stage and a finite element calculation. At the preparatory stage, the model is imported or created from scratch in the Abaqus/CAE pre-processor.

The first modules “Part”, “Property”, “Assembly”, “Mesh” are responsible for the preparatory stage of setting properties, parameters, and mesh before static analysis. At the “Step” module, the stage of the study, its parameters and a set of output data from the calculation results are set. After the calculation step is created,

loads and boundary conditions are set in the “Load” module. In the main module “Optimization”, the appropriate type of optimization task is selected first. After that the Design Response Editor creates and configures responses for the model area that will be used by the target functions and constraints.

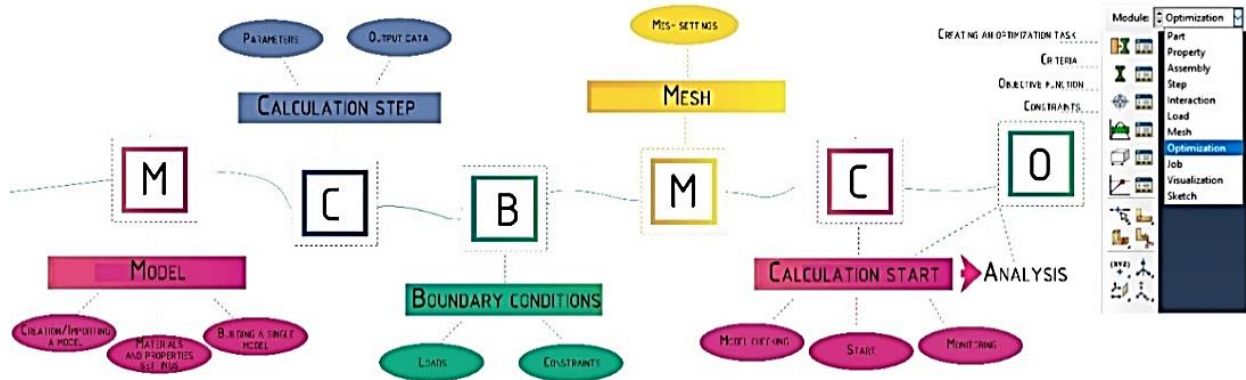


Figure 4. The process of shape optimization in Simulia Abaqus

The goal of optimization is always an objective function, so the next step after determining the design responses is to select them with an indication of the goal: maximizing, minimizing, or minimizing the maximum values. Next, restrictions are specified and to configure them a project response or a combination of responses must be selected and the value of the constraint must be specified. It is also possible to set geometric constraints and define frozen areas to set the boundaries of changes that the module can make during the calculation. To optimize the shape, a local installation condition is available as a parameter that will indicate the moment when optimization is completed.

After completing all the above tasks, the optimization process is started and transferred to the Simulia Tosca Structure pre-processor, which can be restarted if necessary.

Rhinoceros+Karamba. Rhinoceros 7 (Rhino 7) is a 3D modeling software package in which it is possible to create, edit, analyze and document three-dimensional models, as well as perform their visualization and animation. All surfaces in Rhino are free-form surface models created using the curve modeling method (NURBS or Non-Uniform Rational Basis Spline). The NURBS belong to the category of parametric curves and surfaces, and they become the standard for describing and modeling free-form shapes in computer-aided design (CAD) and computer graphics [37]. It should be noted that Rhino is not only a simulation software, but it is also possible to perform calculations in it. The special visual plugin Grasshopper is responsible for the computational process, which is a graphical algorithm editor that runs in conjunction with Rhino. Grasshopper has a connection with the 3D tools of the software and makes it possible to perform all kinds of tasks at a fairly good level due to a large number of additional components. These components provide an opportunity to quickly and efficiently perform the necessary calculations, in particular including optimization. The connection between Grasshopper and Rhino provides the basis for the development of high-level software logic. Depending on the task itself and the shape under study, you can choose the application that is suitable for simulation and optimization. Among the well-known optimization plugins in Grasshopper, the following can be mentioned: Ameba for topological optimization based on the method of bidirectional evolutionary structural optimization (BESO); Octopus for multi-purpose evolutionary optimization based on the Pareto principle; Galapagos and Opossum for single-purpose optimization; Kangaroo for interactive modeling and for performing optimization in real time (Figure 5).

Also, there is an application for structural analysis and optimization for Grasshopper such as Millipede, which makes it possible to perform linear elastic analysis of frame and shell elements in 3D or 2D plate elements. All created algorithms are based on the topology optimization method, which is built into the application. Due to the good data processing speed, Millipede can be used in combination with the Galapagos plugin, which implements evolutionary algorithms to solve common problems of form search. For example, an architectural design project was created for the roof of the Azadi Stadium (architect Arash Naderpour) with the help of Millipede and Galapagos. It was performed as part of a course on architectural technology at the Tehran University of Arts (Figure 6).¹ The roof of the stadium was created using parametric modeling. To optimize the deflection and weight of the structure, the Millipede plugin was implemented.

¹ Parametric design & structural optimization of Azadi stadium roof. Available from: <https://vimeo.com/119049707> (accessed: 12.12.2022).

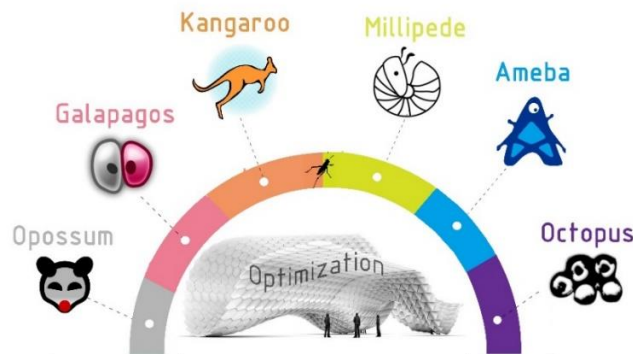


Figure 5. Plugins for optimization in Grasshopper

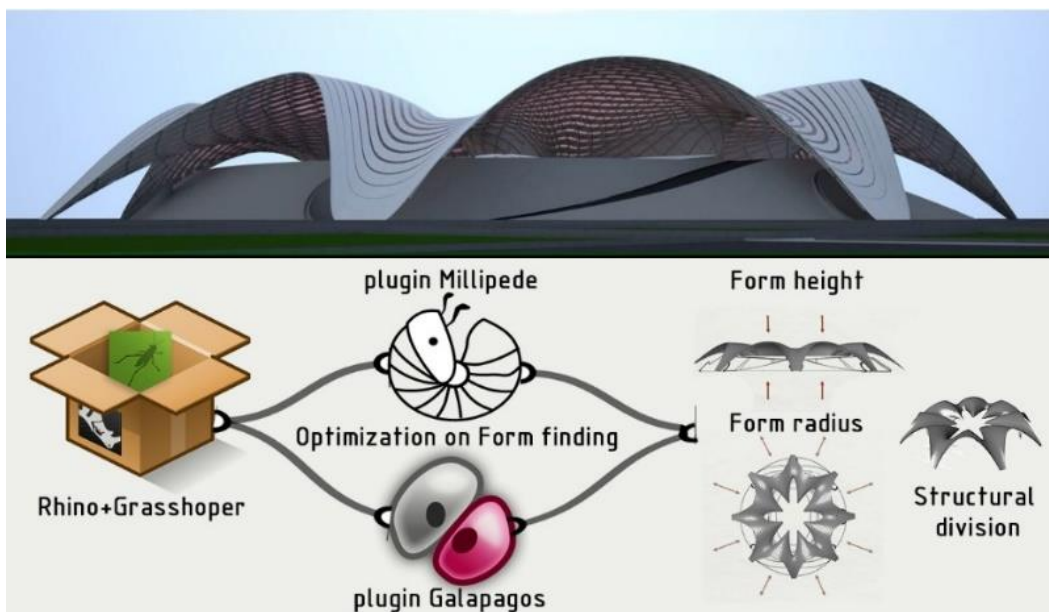


Figure 6. Optimization of the Azadi Stadium roof shape in Rhino + Grasshopper

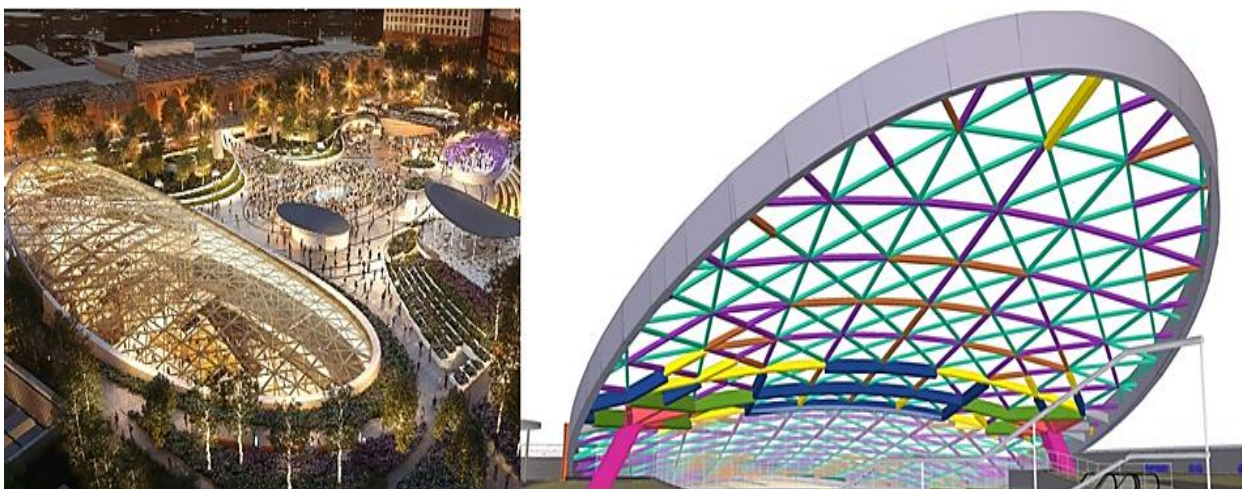


Figure 7. The shape of a large dome in the shopping center "Paveletskaya Plaza"

In conjunction with Galapagos, Karamba 3D application is also widely used, based on the finite element method and being a tool for parametric design of structures [38]. With the collaboration of Galapagos and Karamba 3D, it is possible to achieve the optimal solution for the shape under study quite efficiently and quickly.

As an example, these two applications were used for the technical process of designing a large dome in the shopping center “Paveletskaya Plaza” (Moscow, Russia, APEX Bureau) (Figure 7).

The search for the optimal arch shape was carried out in Karamba 3D, and the search for the optimal combination of sections in the dome was made in Galapagos (Figure 8). The minimum mass and the number of sections (at least 8 per dome) were introduced as criteria. Thus, due to the application of the evolutionary algorithm in Galapagos, the selection of the optimal stiffness ratio and the reduction in the weight of the structure by 40% were implemented compared to the traditional method of searching for sections in such software packages as Autodesk Robot, LIRA.²

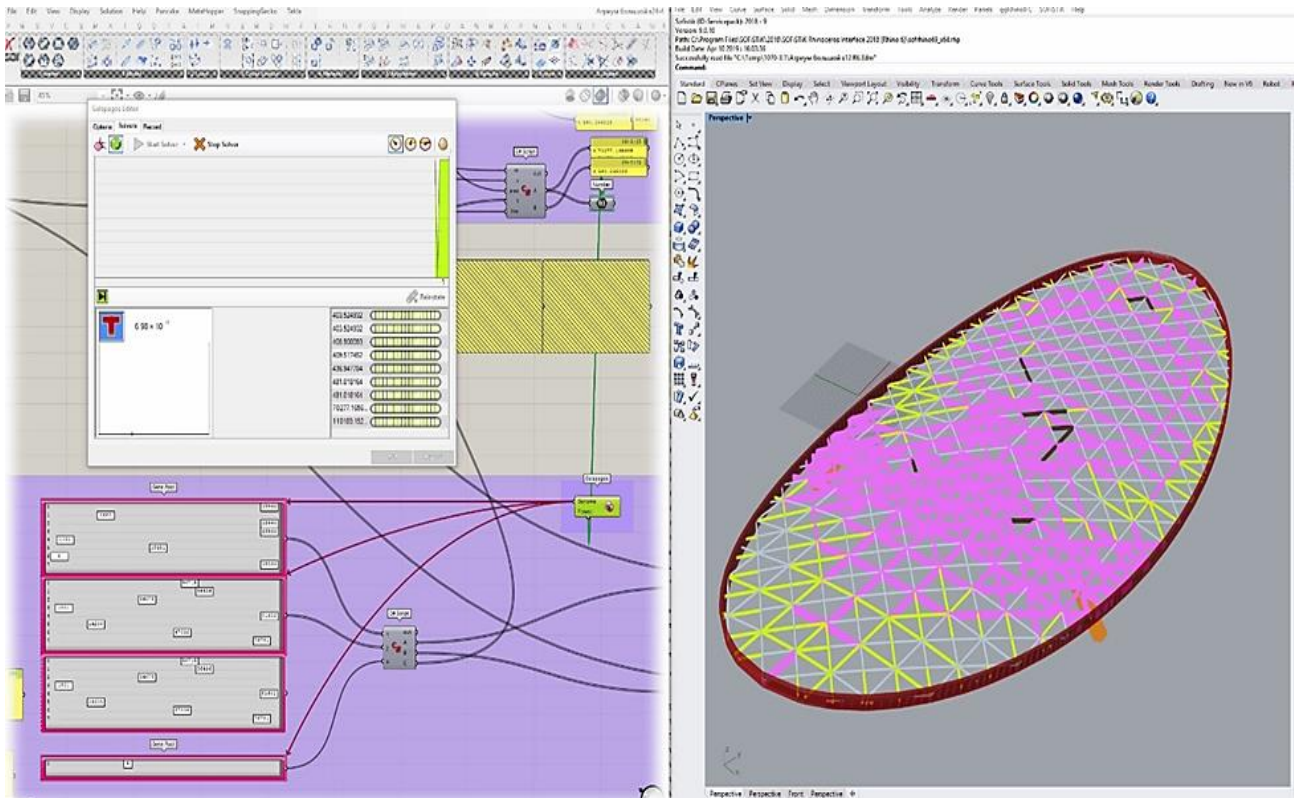


Figure 8. Using the Galapagos solver to optimize sections in the dome

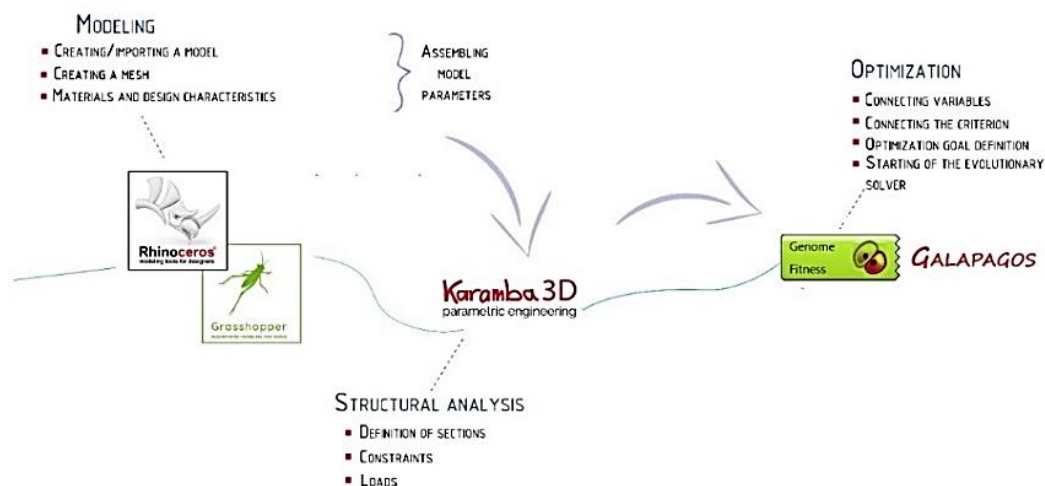


Figure 9. The general optimization algorithm in Rhino and Grasshopper with Karamba and Galapagos

² Design automation with Tekla structures: APEX. Available from: <https://www.tekla.com/ru/опыт/ресурсы/автоматизация-проектирования-с-tekla-structures-опыт-апек> (accessed: 08.12.2022).

The general optimization process in conjunction with Rhino and Grasshopper together with Karamba and Galapagos is shown in Figure 9.

Grasshopper input creates a visual model in Rhino space and all changes that occur with the model are reflected in real time. After creating/importing the main geometry of the model, the materials and the computational mesh are set. To carry out structural analysis through Grasshopper, Karamba 3D application is selected, an analytical model is formed and the calculation is carried out by the finite element method.

To optimize the shape, Galapagos plugin is used, which utilize Karamba output data and geometric characteristics to change the parameters and find the most suitable solution. The plugin is based on a genetic algorithm [39] using evolutionary methods. Genetic algorithms are often applied in the design of capital construction projects of complex geometry, individual architectural elements, during selection of the most successful variants of apartment layouts when deciding on the location and orientation of the building on the site, etc. [40].

Galapagos creates a set of solutions based on genomes (variables to be changed) that respond to fitness (a desired criterion that needs to be minimized or maximized). Thus, to start the optimization process, it is necessary to connect Galapagos Genome to variables and one criterion (function) Galapagos Fitness Input. Then there is a transition to the Galapagos editor, where the optimization goal is selected: maximizing or minimizing the selected parameter. As a result, an effective solution is found that allows to save suitable genomes for a generation and then connect them with other favorable genomes in the next generation.

3. Results and discussion

All the considered software packages have quite powerful functionality for shape optimization and are widely used in the framework of design, calculation, educational and scientific activities. However, not all of them are distributed freely for educational purposes and should have special licenses. An example is COMSOL Multiphysics. The remaining software packages are provided with free licenses for use for educational purposes, except the Rhinoceros with Grasshopper, where only a paid license is provided, but with the capability of purchasing it at a discounted rate. The presence of a Russified version in the official version of the software packages is not provided.

The user interface is one of the most important parts of any software package since it is with it that the user interacts in the process of his work. COMSOL, Ansys, Abaqus and Rhinoceros with Grasshopper provide the principle of structuring the content. COMSOL Multiphysics uses a convenient Model Builder manager, which contains all the necessary data for completion, and in Abaqus, the solution stages are determined by special modules included in the Abaqus/CAE pre-postprocessor and sequentially located under each other. Modules provide fast filling of information, which is then reflected in the task model tree. Since all data is collected in the model tree, it is a fairly complete structure in which all the functions contained in Abaqus are collected. Therefore, for novice users, working with the model tree may be difficult due to the abundance of information. With Ansys, the block diagram provides insight into engineering intent, relationships between data, and the status of an analysis project at a glance. The principle of the block diagram is also implemented in Grasshopper, however, due to a large number of functions and the need to download new plugins to solve specialized tasks, the process of understanding and implementing the solution can take a long time. Table 2 shows a comparison of the software packages by the availability of a training and Russified version, by system requirements and by the user interface.

At the preparatory stage, a convenient function is an ability to build a model directly in the calculation software itself in a special module. Also, importing a model from another CAD system does not cause problems.

Before proceeding to the optimization process, certain parameters and characteristics of the model are set. Also, it is necessary to note the capabilities of software packages for constructing a computational mesh, which is one of the most important stages before the calculation. The computational time of the result depends on the quality of the mesh. COMSOL Multiphysics has the largest selection of ready-made mesh sizes and therefore in most cases, it is not necessary to set the dimensions manually.

In Ansys Workbench, there are fewer preset mesh sizes, whereas, in Abaqus, the mesh is built mainly by setting parameters, although an automatic mesh can also be built from scratch. The same process of building a mesh occurs in Rhino with Grasshopper. Meshes in Rhinoceros consist of triangles and quadrilaterals. Table 3 shows a comparison of software packages for the capability of building a mesh.

Each software package is responsible for carrying out the optimization process in its special module with a solver, which includes a certain set of data to fill in. Since there are currently many optimization methods, those software packages where it is possible to solve the problem using several methods and computational algo-

rithms get a great advantage. It is possible to mention COMSOL Multiphysics and Ansys Workbench, in which more than one method can be used during a computational experiment, which makes it possible to choose the most optimal solution.

The availability of a library of materials with set values of characteristics is also an important factor affecting the convenience of the calculation, which is fully implemented in COMSOL and Ansys. The Abaqus software is dominated by manual input of materials with their properties. A comparison of software packages for the optimization process is given in Table 4.

Table 2

Capabilities of software packages according to system requirements, interface and special versions

Software package	Educational version	Russified version	Interface	
			Convenience of work	Simplicity of the interface
COMSOL Multiphysics 6.1	–	–	+	+
Ansys Mechanical 2022/R1	+	–	+	+
Simulia Abaqus 2022	+	–	+	±
Rhinoceros 7	+	–	+	±

Table 3

Capabilities of building a mesh in software packages

Software package	Mesh		Mesh settings	
	Number of preset mesh sizes	Geometric mesh type	Automatic	Custom
COMSOL Multiphysics	9 (from extremely fine to extremely coarse)	Tetrahedra, hexahedra, prisms and pyramids	+	+
Ansys Workbench	3 (coarse, medium and fine)		+	+
Abaqus	–	Tetrahedra, hexahedra, prisms	+	+
Rhino with Grasshopper	–	Triangles and quadrilaterals	+	+

Table 4

Capabilities of the shape optimization process in software packages

Software packages	Module	Number of optimization methods	Amount of library data		Additional software, plugin or module for shape optimization
			Materials library	Objective library	
COMSOL	Shape Optimization	3	+	+	–
Ansys Workbench	Structural Optimization	3	+	+	+
Abaqus	SIMULIA Tosca Structure Shape	1	–	+	+
Rhino & Grasshopper	Galapagos	1	+	–	+

4. Conclusion

According to the results of the study, the following conclusions can be made:

1. Although there is a currently increasing interest in structural optimization and its types, only several software packages allow providing shape optimization calculations. COMSOL Multiphysics 6.0, Ansys Mechanical 2022/R1 and Simulia Abaqus 2022 are currently the main calculation software packages in which the shape optimization process is implemented. The capability of shape optimization in modeling software packages is still at the stage of development, and so far the solution to this type of tasks is fully implemented only in Grasshopper special visual plugin for Rhino.

2. The presence of a special optimization module in the functionality of software packages expands the possibilities for solving problems of finding the optimal shape. It also speeds up the computational process due to the choice of a mathematical algorithm embedded in the software, the number of iterations, constraints, functions, and characteristics of the model.

3. All the considered software packages have an accessible, structured user interface. All model data in COMSOL and Abaqus is contained in a special manager (the model tree of the project), whereas in Rhino with Grasshopper and Ansys, a block data type is implemented, where all tasks are in separate interconnected blocks.

4. According to the number of functional capabilities in the process of shape optimization, Comsol Multiphysics can be noted for its wide range of optimization methods and preset mesh sizes.

5. Ansys Mechanical has the most extensive database of materials and properties in terms of the amount of library data.

6. As part of educational access, software packages such as Ansys and Abaqus are available, which provide free educational licenses and learning platforms, but there is a limit on the estimated number of mesh elements.

7. To work with all studied software packages knowledge of English is required, since there is no possibility of choosing a Russian-language version in them.

The final decision on the choice of a suitable software package for shape optimization should be made taking into account all of the above characteristic features of the software packages.

References

- Allaire G., Dapogny C., Jouve F. Shape and topology optimization. In: Bonito A., Nochetto R.H. (eds.) *Geometric Partial Differential Equations. Part II. Handbook of Numerical Analysis*. 2021;(22):1–93.
- Theodossiou N., Kougias I., Karakatsanis D. The history of optimization. Applications in water resources management. *IWA Regional Symposium on Water, Wastewater and Environment: Traditions and Culture*. Patras; 2014. p. 345–355.
- Mykel J., Kochenderfer, Tim A. *Wheeler algorithms for optimization*. London: The MIT press; 2019.
- Rao S.S. *Engineering optimization: theory and practice*. Hoboken, New Jersey: John Wiley & Sons; 2009.
- Dede T., Kripka M., Togan V., Yepes V., Rao V.R. Usage of optimization techniques in civil engineering during the last two decades. *Current Trends in Civil & Structural Engineering*. 2019;2(1):1–17. <https://doi.org/10.33552/CTCSE.2019.02.000529>
- Christensen P.W., Klarbring A. *An introduction to structural optimization*. Springer Science, Business Media B.V.; 2009.
- Mei L., Wang Q. Structural optimization in civil engineering: a literature review. *Buildings*. 2021;11(2):1–27. <https://doi.org/10.3390/buildings11020066>
- Shevtsov S.N. *Methods for optimizing structures*. Rostov-on-Don; 2010. (In Russ.)
Шевцов С.Н. Методы оптимизации конструкций. Ростов н/Д., 2010. 20 с.
- Srivastava P.K., Simant S.S. Structural optimization methods: a general review. *International Journal of Scientific Research & Engineering Trends*. 2017;6(9):88–92.
- Deaton J.D., Grandhi R.V. A survey of structural and multidisciplinary continuum topology optimization: post 2000. *Structural and Multidisciplinary Optimization*. 2014;49:1–38. <https://doi.org/10.1007/s00158-013-0956-z>
- Guest J.K., Lotfi R., Gaynor A., Jalalpour M. Structural topology optimization – moving beyond linear elastic design objectives. *Structures Congress 2012*. Chicago, Illinois; 2012. p. 245–256.
- Hinz M., Magoulès F., Rozanova-Pierrat A., Rynkovskaya M., Teplyaev A. On the existence of optimal shapes in architecture. *Applied Mathematical Modelling*. 2020;(94):676–687. <https://doi.org/10.48550/arXiv.2010.01832>
- Hinz M., Rozanova-Pierrat A., Teplyaev A. Boundary value problems on non-Lipschitz uniform domains: stability, compactness and the existence of optimal shapes. *Asymptotic Analysis*. 2023;(Pre-press):1–37. <https://doi.org/10.3233/ASY-231825>
- Haslinger J., Mäkinen R.A.E. *Introduction to shape optimization: theory, approximation, and computation*. Philadelphia: SIAM; 2003.
- Allaire G., Henrot A. On some recent advances in shape optimization. *Comptes Rendus de l'Académie des Sciences-Series IIB-Mechanics*. 2001;329(5):383–396. [https://doi.org/10.1016/S1620-7742\(01\)01349-6](https://doi.org/10.1016/S1620-7742(01)01349-6)
- Slawig T., Prüfert U. Mathematics-based optimization in the COMSOL Multiphysics framework. *Proceedings of the COMSOL User Conference*. Stuttgart; 2011. p. 1–6.
- Hashim A.A., Mahmoud K.I., Ridha H. Geometry and shape optimization of piezoelectric cantilever energy harvester using COMSOL Multiphysics software. *International Review of Applied Sciences and Engineering*. 2021;12(2):103–110. <https://doi.org/10.1556/1848.2021.00170>
- Cui G. Shape optimization based on ANSYS. *Journal of Information and Computational Science*. 2015;12(11):4291–4297.
- Seranaj A., Elezi E., Seranaj A. Structural optimization of reinforced concrete spatial structures with different structural openings and forms. *Research on Engineering Structures and Materials*. 2018;(4):79–89. <https://doi.org/10.17515/resm2016.79st0726>
- Tomás A., Martí P. Shape and size optimization of concrete shells. *Engineering Structures*. 2010;32(6):1650–1658. <https://doi.org/10.1016/j.engstruct.2010.02.013>
- Husainie S.N. Weight optimization of a landing gear steering collar using Tosca in Abaqus. *Science in the Age of Experience*. Boston, MA; 2016. <https://doi.org/10.13140/RG.2.2.33591.32163>

22. Omid H., Golabchi, M. Survey of parametric optimization plugins in Rhinoceros used in contemporary architectural design. *Proceedings of the Fourth International Conference on Modern Research in Civil Engineering, Architecture, Urban Management and Environment*. Karaj; 2020. p. 1–9.
23. Boon C., Griffin C., Papaefthimious N., Ross J., Storey K. Optimizing spatial adjacencies using evolutionary parametric tools. *Rerkins+Will Research Journal*. 2015;(07.02):24–37.
24. Cubukcuoglu C., Ekici B., Tasgetiren M.F., Sariyildiz S. OPTIMUS: self-adaptive differential evolution with ensemble of mutation strategies for grasshopper algorithmic modeling. *Algorithms*. 2019;12(7):141. <https://doi.org/10.3390/a12070141>
25. Dai R., Kerber E., Brell-Cokcan S. Robot assisted assembly of steel structures: optimization and automation of plasma cutting and assembly. *CAAD – Computer-Aided Architectural Design Research in Asia*. Wellington; 2019. <https://doi.org/10.52842/conf.caadria.2019.1.163>
26. Kovtun V.A., Korotkevich S.G. Review of current application programs for research composite products. *Technology of Technosphere Safety*. 2016;(1):1–9. (In Russ.)
Ковтун В.А., Короткевич С.Г. Обзор современных прикладных программных комплексов для проведения исследований композитных изделий // Технологии техносферной безопасности. 2016. № 1 (65). С. 1–9.
27. Choi W., Huang C., Kim J., Park G. Comparison of some commercial software systems for structural optimization. *11th World Congress on Structural and Multidisciplinary Optimisation*. Sydney; 2015. p. 1–6.
28. *Structural mechanics module. User's guide*. Available from: <https://doc.comsol.com/5.5/doc/com.comsol.help.sme/StructuralMechanicsModuleUsersGuide.pdf> (accessed: 14.12.2022).
29. Burkova E.N., Kondrashov A.N., Rybkin K.A. *COMSOL automated calculation system*. Perm: Perm State National Research University; 2019. (In Russ.)
Буркова Е.Н., Кондрашов А.Н., Рыбкин К.А. Система автоматизированных расчетов COMSOL. Пермь: Пермский государственный национальный исследовательский университет, 2019. 133 с.
30. Kurushin A.A. *Solution of multiphysics microwave problems using CAD COMSOL*. Moscow: One-Book; 2016. (In Russ.)
Курушин А.А. Решение мультифизических СВЧ задач с помощью САПР COMSOL. М.: One-Book., 2016. 166 с.
31. Ermakova E., Elberdov T., Rynkovskaya M. Shape optimization of a shell in COMSOL multiphysics. *Computation*. 2022;10(4):54. <https://doi.org/10.3390/computation10040054>
32. Rynkovskaya M.I., Elberdov T., Sert E., Öchsner A. Study of modern software capabilities for complex shell analysis. *Structural Mechanics of Engineering Constructions and Buildings*. 2020;16(1):45–53. <https://doi.org/10.22363/1815-5235-2020-16-1-45-53>
33. Proshin M.V., Isaikova T.V., Kabanov Y.Y. Using ANSYS/ CivilFEM to solve building design problems. Overview of the possibilities of the software complex. *Rational Management of the Enterprise*. 2008;(2):32–34. (In Russ.)
Прошин М.В., Исайкова Т.В., Кабанов Ю.Ю. Использование ANSYS/CivilFEM для решения задач строительного проектирования. Обзор возможностей программного комплекса // Рациональное управление предприятием. 2008. № 2. С. 32–34.
34. Zhidkov A.V. *Application of the ANSYS system to solving problems of geometric and finite element modeling: educational and methodological material for the advanced training program "Information systems in mathematics and mechanics"*. Nizhny Novgorod; 2006. (In Russ.)
Жидков А.В. Применение системы ANSYS к решению задач геометрического и конечно-элементного моделирования: учебно-методический материал по программе повышения квалификации «Информационные системы в математике и механике». Н. Новгород, 2006.
35. Kuznetsova T.A. Beams analysis using "Abaqus" software package. *Vestnik VGAVT*. 2016;(47):209–220. (In Russ.)
Кузнецова Т.А. Расчет балок с помощью программного комплекса Abaqus // Вестник ВГАВТ. 2016. № 47. С. 209–220.
36. Oganesyanyan P.A., Shevtsov S.N. Topology designs optimization in ABAQUS package. *Izvestia of Samara Scientific Center of the Russian Academy of Sciences*. 2014;16(6–2):543–549. (In Russ.)
Оганесян П.А., Шевцов С.Н. Оптимизация топологии конструкций в пакете ABAQUS // Известия Самарского научного центра Российской академии наук. 2014. Т. 16. № 6–2. С. 543–549.
37. Sassone M., Pugnale A. Evolutionary structural optimization in shells design. *Advanced Numerical Analysis of Shell-like Structures*. 2007;(4):247–257.
38. Rumph M. *Informed form generation-embedding simulation and optimization into architectural design* (dissertation). Kassel; 2018. <https://doi.org/10.17170/kobra-202012162602>
39. Jakiela M.J., Chapman C., Duda J., Adewuya A., Saitou K. Continuum structural topology design with genetic algorithms. *Computer Methods in Applied Mechanics and Engineering*. 2000;186(2–4):339–356. [https://doi.org/10.1016/S0045-7825\(99\)00390-4](https://doi.org/10.1016/S0045-7825(99)00390-4)
40. Knyazeva N.V. Application of evolutionary algorithms for automated routine tasks of enumerating design options. *Engineering and Construction Bulletin of the Caspian Sea*. 2021;(3):73–77. (In Russ.) <https://doi.org/10.52684/2312-3702-2021-37-3-73-77>
Князева Н.В. Использование эволюционных алгоритмов для автоматизации рутинных задач перебора вариантов проектных решений // Инженерно-строительный вестник Прикаспия. 2021. № 3 (37). С. 73–77. <https://doi.org/10.52684/2312-3702-2021-37-3-73-77>