Syllabus
Course Program

Modeling in CAE Systems

Specialty
113 – Applied Mathematics

Institute
Institute of Computer Modeling, Applied Physics and Mathematics

Educational program
Computer and Mathematical Modeling

Department
Mathematical Modeling and Intelligent Computing in Engineering (161)

Level of education
Master's level (1 year 4 months)

Course type
Special (professional), Mandatory

Semester
1

Language of instruction
English

Lecturers and course developers

Gennadii Martynenko (responsible lecturer)

gennadii.martynenko@khpi.edu.ua

Doctor of Engineering Sciences, Professor, Professor at the Department of Mathematical Modeling and Intelligent Computing in Engineering of NTU “KhPI”.
Scientific and pedagogical work – 18 years. Author of more than 180 scientific and educational works.

Lecturer and laboratory workshop teacher of the discipline:
"Database organization", "Data Mining", “Software systems for design and analysis”, “Software for simulation of physical processes”, “Modeling of objects and processes in CAD/CAE systems”, “Analysis of dynamic processes in CAD/CAE systems”, “Modeling in CAE systems”, “Approximate and numerical methods for solving nonlinear problems”, "Pedagogical and information technologies in applied mathematics".
More about the lecturer on the department's website

Lyudmyla Rozova (assistant, laboratory practical teacher)

lyudmyla.rozova@khpi.edu.ua

Candidate of Technical Sciences (PhD), Associate Professor, Associate Professor at the Department of Mathematical Modeling and Intelligent Computing in Engineering of NTU “KhPI”.
Experience in scientific and pedagogical work – 17 years. Author of more than 40 scientific and educational works.

Lectures in the disciplines: “Object-oriented programming”, “Modeling of multiphysics processes”, “Object Oriented Programming (English)”, “Data analysis in Python and R”; laboratory workshops in the disciplines: “Object-oriented programming”, “Programming”, “Modeling in CAE systems”
More about the lecturer on the department's website

General information

Summary

The discipline is aimed at developing the knowledge, skills and abilities necessary for the practical use of modern multi-purpose design and analysis packages, namely CAE (Computer-Aided Engineering) the ANSYS Workbench finite element analysis system, in order to solve scientific and applied problems in the field of professional and research innovation activities, mastery of modern world trends in the development of methods for computer modeling of objects, namely structures or their elements, and specialized finite analysis of processes in structures, namely structural strength, thermal conductivity and thermoelasticity, natural and free vibrations, mono- and multibody dynamics, coupled thermal and structural strength static, quasi-static and dynamic phenomena, in sufficient volume for use in practical professional activities. All stages of each structural analysis are considered, namely the theoretical foundations and algorithms for performing analyzes of static structural strength, thermal conductivity and thermoelasticity, vibrations, multibody dynamics and coupled fields, including elements of geometric modeling, types of finite elements, meshing methods using various problem statements and corresponding them types of finite elements, mesh quality indicators and mesh refinement, construction of a calculation model with boundary conditions and loads of different types and taking into account various types of symmetry, problem solving methods and solvers, output of calculation results, assessment of the quality of the finite element model and calculation results using various criteria. The possibilities of performing analyzes using ANSYS Workbench automation using the Python API, namely the software library of functions and classes (set of components) PyMAPDL (Python Mechanical APDL), are separately considered.

Course objectives and goals

The goals of the discipline is: to develop students' knowledge of existing modern approaches to the methods, methodologies and techniques of theoretical analysis and practical application of computer CAE systems of engineering design and analysis for constructing physical models of objects, selection of theoretical foundations for the technical specifications and formulation of the problem and solution method and the corresponding analysis module, construction of geometric models of structures or their elements, formation of finite element calculation models with assessment of their quality and analysis of the results of calculation studies when solving structural problems of static strength, thermal conductivity and thermoelasticity, natural vibrations, multibody dynamics and coupled fields under various load configurations, boundary conditions and symmetry conditions.
The objectives of teaching the discipline are: providing students with in-depth knowledge of methods and programs for solving problems of static strength, thermal conductivity and thermoelasticity, natural vibrations, multibody dynamics and coupled fields for structures and their elements; training in working with a specialized software package for modeling and final element analysis of processes ANSYS Workbench; mastering the process of solving problems, which consists of constructing physical models of real objects, geometric modeling, creating computational models, setting up the solution and the solution itself, outputting the solution in graphical and text form, assessing the accuracy of numerical results and their analysis with checking working conditions depending on type of analysis. When solving most of these problems, the main labor involved is determining the parameters that characterize the state of the object depending on the formulation of the problem and the analysis performed, therefore increased attention is paid to solving problems based on the finite element method. At the same time, methods are considered for reducing the dimensionality of problems through the use of various types of finite elements, as well as through the use of lumped factors, taking into account the plane and axial symmetry of systems and loads. The listed methods and techniques for studying the parameters of structures and mechanical systems are demonstrated by solving specific problems often encountered in practice, using the interactive mode of operation of the ANSYS Workbench software package and automation of ANSYS Workbench using a software library of functions and classes or a set of PyMAPDL components, which is included in PyAnsys - the collection of many Python packages for using ANSYS products through Python.

Format of classes

Lectures, laboratory classes, consultations, self-study. Final control in the form of an exam.

Competencies

Program competencies according to the educational program:
GC3: Ability to master modern knowledge, formulate and solve problems;
GC7: Ability to think abstractly, analyse and synthesise;
PC1: Ability to solve tasks and problems that can be formalised, require updating and integrating knowledge, in particular in conditions of incomplete information;
PC2: Ability to conduct scientific research aimed to develop new and adapt existing mathematical and computer models to study various processes, phenomena and systems, conduct appropriate experiments and analyse the results;
PC3: Ability to develop methods and algorithms for the construction, research and software implementation of mathematical models in engineering, physics, biology, medicine and other fields and to analyse them;
PC4: Ability to develop and research mathematical and computer models, conduct computational experiments and solve formalised problems using specialised software;
PC9: The ability to mathematically formalise the formulation of scientific and practical problems, to choose a mathematical analytical or numerical method of its solution, which ensures the required accuracy and reliability of the result;
PC11: Ability to mathematically describe various dynamic processes that can occur in systems of design objects;
PC12: Ability to identify the essence of scientific and technical problems in professional activities, to apply appropriate mathematical models for the study of mechanical objects and processes.

Learning outcomes

Program learning outcomes according to the educational program:
LO4: Build mathematical models of complex systems and choose methods of their research, implement the built models in software and check their adequacy using computer technologies.
LO6: Apply procedures for formal description of systems, checking their adequacy for the study of socio- economic, technical, natural and other systems.
LO9: Be able to analyse and design systems with large amounts of data, apply and adapt methods of knowledge acquisition, methods of evaluation and interpretation of the found patterns.
LO11: Possess skills of abstract thinking, analysis and synthesis.
LO14: To have the knowledge to mathematically formalise the formulation of scientific and practical problems, to choose a mathematical analytical or numerical method of its solution, which ensures the required accuracy and reliability of the result.
LO15: To be able to carry out mathematical and computer modelling, computational experiment, solve formalised problems using specialised software.
LO17: Possess knowledge of the mathematical description of various dynamic processes that can occur in systems of design objects.

Student workload

The total volume of the course is 120 hours (4 ECTS credits): lectures - 32 hours, laboratory classes - 16 hours, self-study - 72 hours.

Course prerequisites

The study of the course is based on the knowledge gained during the completion of the bachelor's educational program in specialty 113 – Applied Mathematics, educational program - Computer and Mathematical Modeling (concept, formulations, approaches and methods of programming, mathematical analysis, modeling and mechanics of solid deformable bodies), and on information, considered in the disciplines of the curriculum:
PP 1. Methods of mathematical modeling and data analysis;
PP 2. Nonlinear processes and models; 4.1.1.2. Mixed problems for thin-walled structures;
4.1.1.3. Elastic-plastic deformation of plates and shells;
4.1.2.1. Mathematical methods of analysis of machine dynamics
4.1.2.2. Computer modeling of dynamics and vibration protection of rotary machines;
4.1.3.1. FEM algorithms; 4.1.3.2. Programming of modern numerical methods.

Features of the course, teaching and learning methods, and technologies

The course "Modeling in CAE systems" (1st semester) consists of 15 training sessions, each of which has 3 components - lecture, independent work and laboratory practice, as well as one final session. As part of the course, lectures are conducted interactively using multimedia technologies. In laboratory classes, a practically oriented approach to training is used, general and individual tasks are performed, allowing one to gain knowledge and skills in the use of modern software tools and applications for modeling quasi-static and dynamic linear physical processes in structures and their elements to assess the static and dynamic strength of these objects with taking into account the thermal state and field coupling, as well as gain knowledge and skills to automate the simulation of these processes using existing specialized software libraries of functions and classes or sets of components for the Python programming language.

Program of the course

Topics of the lectures

MODULE #1. Lecture classes (LCs) 2 credits / 32 hours "THEORETICAL BASIS OF MODELING QUASI-STATIC AND DYNAMIC PROCESSES IN STRUCTURES IN CAE SYSTEM ANSYS WORKBENCH"

Topic 1. Static structural finite element analysis in ANSYS Workbench.

Introduction to modeling in CAE systems, including the ANSYS Workbench:1. Introduction to the finite element method (general, history, main components of FEM software). 2. Concept (physical model, discretization, degrees of freedom, types of elements, linear and nonlinear FE models, systems of units). 3. ANSYS Workbench graphical user interface (GUI).

Topic 2. Static structural finite element analysis in ANSYS Workbench.

Beam, link, plate and shell elements in ANSYS Workbench: 1. Modeling of one-dimensional structures. 2. Trusses (element: LINK180). 3. Beams (elements: BEAM188 and BEAM189). 4. Modeling of two-dimensional structures. 5. Membranes and shells (elements: SHELL181 and SHELL281). 6. ANSYS Workbench graphical user interface (GUI).

Topic 3. Static structural finite element analysis in ANSYS Workbench.

Plane elements in ANSYS Workbench: 1. Plane problems of the theory of elasticity. 2. Plane elements (plane strain and plane stress assumptions, elements: PLANE182 and PLANE183). 3. Key options – Keyoptions. 4. Saving the project. 5. Create a 2D plane strain or plane stress simulation.

Topic 4. Static structural finite element analysis in ANSYS Workbench.

Volume elements in the ANSYS Workbench: 1. Modeling of 3D bodies with a general formulation of the problem. 2. Comparison of volumetric solid and finite element models. 3. Approaches to creating a volumetric solid model. 4. Volumetric elements (elements: SOLID185, SOLID186 and SOLID187). 5. Surface effects elements (elements: SURF153, SURF154 and SURF156). 6. Creation of 3D simulation of rigid body behavior.

Topic 5 Static structural finite element analysis in ANSYS Workbench.

Meshing in ANSYS Workbench: 1. Mesh quality. 2. Named selections. 3. Mesh Metric (mesh quality indicators). 4. Mesh Method (setting the mesh creation method). 5. Overview of the mesh from the inside. 6. Integration schemes.

Topic 6. Static structural finite element analysis in ANSYS Workbench.

Mesh refinement in ANSYS Workbench: 1. Mesh independence/convergence. 2. Stress singularity and stress concentration. 3. Size Functions (size determination functions). 4. Pinch. 5. Inflation. 6. Refinement of finite element solutions.

Topic 7. Static structural finite element analysis in ANSYS Workbench.

Solvers in ANSYS Workbench: 1. Solvers. 2. Explicit and implicit solvers. 3. Implicit solver solution strategy. 4. Explicit dynamics solver solution strategy. 5. Solver example: explicit solver. 6. Solver example: implicit solver. 7. Solver example: Explicit vs. Implicit solver. 8. Convergence. 9. Radius of convergence. 10. Direct and indirect solvers. 11. Load steps and substeps.

Topic 8. Static structural finite element analysis in ANSYS Workbench.

Symmetry, periodic boundary conditions (PBCs) and representative volume elements (RVEs) in ANSYS Workbench: 1. Types of symmetry (symmetry, antisymmetry, axial symmetry, cyclic symmetry). 2. Periodic boundary conditions / Representative volume elements.

Topic 9. Thermal analysis of structures in ANSYS Workbench.

Thermal problems in the ANSYS Workbench – stationary and transient processes: 1. Introduction to thermal analysis (heat transfer mode, types of thermal analysis). 2. Thermal elements (elements: PLANE55, PLANE77, SOLID70 and SOLID90). 3. Sources of engineering data. 4. Setting the time-dependent load.
Topic 10. Dynamic analysis of structures in ANSYS Workbench.
Modal analysis in ANSYS Workbench: 1. Modal analysis (motivation, general provisions, equations). 2. Carrying out modal analysis in ANSYS (general provisions, limiting conditions and solver settings, evaluation of results, summary).
Topic 11. Dynamic analysis of structures in ANSYS Workbench.
Coupled analysis in ANSYS Workbench: 1. Coupling thermal and structural analyzes (step-by-step algorithm). 2. Coupling static structural and modal analyzes (step-by-step algorithm). 3. Coupling thermal, static structural and modal analysis (step-by-step algorithm).
Topic 12. Dynamic analysis of structures in ANSYS Workbench.
Multibody dynamics in ANSYS Workbench: 1. Multibody dynamics (introduction to structural dynamics, types of dynamic analysis in ANSYS, main points of multibody dynamics, two types of multibody dynamic analysis, solver for rigid bodies and equation of motion, solver for flexible bodies and equation of motion, inertia, damping, time-dependent loads, representation of bodies and elements, definition and abstraction of connections, initial and connection conditions, solution results, summary). 2. Dynamics of rigid bodies (general provisions, stages of setting up the problem). 3. Transient structural analysis (multibody dynamics of rigid and flexible bodies, general provisions, stages of problem setup).
Topic 13. Output of results and automation of calculations in ANSYS Workbench.
Automating Workbench with Python: 1. Introduction to PyMAPDL (overview of PyMAPDL, installation requirements, installation, using PyMAPDL commands, control errors and warnings). 2. Demonstration (define parameters, run PyMAPDL, create geometry, define material properties, mesh attributes and create mesh, apply loading and boundary conditions, solve modeling problem, exit PyMAPDL). 3. Graphical representation in PyMAPDL. 4. Other useful commands.
Topic 14. Output of results and automation of calculations in ANSYS Workbench.
Postprocessing in ANSYS Workbench and using Python: 1. Postprocessing in Workbench (graphical display of results - postprocessing, settings, examples of using settings, secant plane, charts). 2. Postprocessing in PyMAPDL.
Topic 15. Resume of the course “Modeling in CAE systems”.
Summary of modeling objects and processes in the CAE system ANSYS Workbench: 1. Brief overview of the main points of the course (introduction to FEM and ANSYS Workbench, static analysis of structures, element types, meshing, mesh refinement, solvers, symmetry, periodic boundary conditions and representative elements volume).
Topic 16. Resume of the course “Modeling in CAE systems”.
Summary of modeling objects and processes in the CAE system ANSYS Workbench: 1. Brief overview of the main points of the course (thermal problems: steady and transient processes, modal analysis, coupled analysis, multibody dynamics, Workbench automation with Python, postprocessing with Python).

Topics of the workshops

There are no classes.

Topics of the laboratory classes

MODULE #2. Laboratory practical classes (LPCs) 1 credit/16 hours “COMPUTER MODELING OF STRUCTURE ELEMENTS AND CARRYING OUT STATIC AND DYNAMIC ENGINEERING ANALYSIS IN ANSYS WORKBENCH” (one class covers 2 topics in 2 hours)
Topic 1. Static structural finite element analysis in ANSYS Workbench.
Task 1 – Static structural analysis of volume structure under various load configurations and boundary conditions in ANSYS Workbench.
Task 2 – Calculation of the values of given physical quantities in various unit systems, using formulas and the ANSYS Workbench.
Topic 2. Static structural finite element analysis in ANSYS Workbench.
Task 1 – Static structural analysis of a beam structure under concentrated loads with output and analysis of results in ANSYS Workbench.
Task 2 – Static structural analysis of a beam structure with a complex cross-section under distributed loads, comparing graphs of results with the analytical solution in ANSYS Workbench.

Task 3 – Static structural analysis of a truss structure with different section sections under distributed loads in ANSYS Workbench.
Task 4 – Comparative structural analysis of the behavior of a thin-walled structure when using membrane and plate (shell) elements in ANSYS Workbench.
Task 5 – Static structural analysis of a complex curved shell structure under distributed loads in ANSYS Workbench.
Task 6 – Static structural analysis of a complex curved shell structure under step-by-step loading in ANSYS Workbench.
Topic 3. Static structural finite element analysis in ANSYS Workbench.
Task 1 – Static structural analysis of a plane structure of a simple shape under a plane stress state with strict restrictions on the shape and size of the elements used in ANSYS Workbench.
Task 2 – Static structural analysis of a plane structure of simple shape under plane stress/strain state under various boundary conditions in ANSYS Workbench.
Task 3 – Static structural analysis of a complex plane structure under plane stress conditions under various boundary conditions in ANSYS Workbench.
Topic 4. Static structural finite element analysis in ANSYS Workbench.
Task 1 – Static structural analysis of a volumetric structure of topologically complex shape under various boundary conditions in ANSYS Workbench.
Task 2 – Static structural analysis to simulate uniaxial tensile testing of a bulk specimen made of a standard material in ANSYS Workbench.
Task 3 – Static structural analysis of a volumetric curved structure of complex shape under specified boundary conditions and loads in ANSYS Workbench.
Topic 5. Static structural finite element analysis in ANSYS Workbench.
Task 1 – Static structural analysis of a beam structure with a complex section under various boundary conditions and loads, meshing methods and finite element sizes in ANSYS Workbench, comparing the result with the analytical solution.
Task 2 – Static structural analysis of a volumetric curvilinear structure of complex shape under specified boundary conditions and loads in ANSYS Workbench with the study of various methods for creating a mesh and checking the corresponding mesh quality indicators.
Task 3 – Analytical analysis of the deformation of linear and quadratic finite elements for given options for their connection and load.
Task 4 – Static structural analysis of a volumetric structure of complex shape to find loads corresponding to specified displacements in ANSYS Workbench.
Topic 6. Static structural finite element analysis in ANSYS Workbench.
Task 1 – Static structural analysis of a simple shaped volumetric structure in ANSYS Workbench and study of equivalent stress convergence using mesh refinement for sharp and rounded corners of a square hole.
Task 2 – Static structural analysis of a simple volumetric structure in ANSYS Workbench and study of equivalent stress convergence under concentrated and distributed load application.
Task 3 – Static structural analysis of a volumetric curved structure with stiffeners under various boundary conditions and loads in ANSYS Workbench, examining convergence using several mesh quality metrics.
Task 4 – Static structural analysis of a complex curved volumetric structure under various boundary conditions and loads in ANSYS Workbench, examining convergence using several mesh quality metrics.
Topic 7. Static structural finite element analysis in ANSYS Workbench.
Task 1 – Quasi-static structural analysis of a volumetric structure of complex curvilinear shape with step-by-step dependence on load time in ANSYS Workbench.
Task 2 – Static structural analysis of a three-dimensional structure of a simple curved shape using various direct and iterative solvers in ANSYS Workbench.
Topic 8. Static structural finite element analysis in ANSYS Workbench.
Task 1 – Static structural analysis of a simple shaped plane structure under plane stress/strain state under various boundary conditions in ANSYS Workbench using symmetry conditions.
Task 2 – Static structural analysis to simulate uniaxial tensile testing of a bulk specimen made of a standard material in ANSYS Workbench using symmetry conditions.
Task 3 – Analytical analysis of design models (structure geometry, boundary conditions and loads) from the point of view of the possibility of applying symmetry conditions.

Task 4 – Static structural analysis of a volumetric curved structure of complex shape under specified boundary conditions and loads in ANSYS Workbench using planar, axial and cyclic symmetry.
Task 5 – Analytical comparison of the results of previous problems when solving without and using symmetry conditions.
Topic 9. Thermal analysis of structures in ANSYS Workbench.
Task 1 – Steady-state thermal analysis of a simple shaped flat structure in a plane statement under various thermal boundary conditions and loads in ANSYS Workbench.
Task 2 – Transient thermal analysis of a simple flat structure in a plane statement under various thermal boundary conditions and loads in ANSYS Workbench.
Task 3 – Steady-state thermal analysis of a three-layer structure made of different materials under various thermal boundary conditions and loads in ANSYS Workbench.
Task 4 – Thermal steady-state analysis of a volumetric spatially branched structure under specified thermal boundary conditions and loads in ANSYS Workbench.
Task 5 – Transient thermal analysis of a simple element of a volumetric structure with different dependencies of thermal boundary conditions and loads on time in different areas in ANSYS Workbench.
Topic 10. Dynamic analysis of structures in ANSYS Workbench.
Task 1 – Modal analysis of a simple shaped flat shell structure under specified boundary conditions in ANSYS Workbench.
Task 2 – Modal analysis of a truss structure in a volumetric statement under various boundary conditions in ANSYS Workbench.
Task 3 – Modal analysis of a volumetric spatially branched structure in the general and cyclosymmetric statements in ANSYS Workbench.
Topic 11. Dynamic analysis of structures in ANSYS Workbench.
Task 1 – Coupled static thermal-strength analysis of a simple structure in the plane statement in ANSYS Workbench.
Task 2 – Coupled thermal-strength transient analysis of a volumetric curved complex structure in ANSYS Workbench.
Task 3 – Coupled thermal-strength and modal analysis of a flat shell structure of simple shape under specified structural boundary conditions and temperature changes in ANSYS Workbench.
Topic 12. Dynamic analysis of structures in ANSYS Workbench.
Task 1 – Multibody dynamics of a roller structure taking into account different types of contacts in ANSYS Workbench.
Task 2 – Multibody dynamics of a simple crank mechanism with the study of the kinematic behavior of elements in ANSYS Workbench.
Task 3 – Multibody dynamics of the crank mechanism of an internal combustion engine with the study of kinematic behavior and deformation of elements in ANSYS Workbench.
Topic 13. Output of results and automation of calculations in ANSYS Workbench.
Task 1 – Automation of static structural analysis of a plate structure in a volumetric statement under specified boundary conditions and kinematic loads in PyMAPDL.
Task 2 – Automation of static structural analysis of a representative volume element with periodic boundary conditions in PyMAPDL.
Topic 14. Output of results and automation of calculations in ANSYS Workbench.
Task 1 – Visualization using various settings of automation results in PyMAPDL of a static structural analysis of a plate structure in a volumetric statement.
Task 2 – Generation and visualization of the strain-stress response as a result of automation in PyMAPDL of the static structural analysis of a representative volume element with periodic boundary conditions.
Topic 15. Resume of the course “Modeling in CAE systems”.
Modular control #1. Computer test (40 random short questions) on engineering analysis of structural elements in ANSYS Workbench.
Topic 16. Resume of the course “Modeling in CAE systems”.
Modular control #2. Presentation and defense of an individual calculation task dedicated to the engineering analysis of structural elements in ANSYS Workbench.

Self-study

1. Basic relationships between the theories of elasticity, thermal conductivity and the theory of thermoelasticity when solving problems of different classes – 12 hours.

2. Finite element method, use of rod, beam, plane and volumetric elements with different functions of shapes – 11 hours.
3. Basic concepts, equations and methods of the theory of free and forced oscillations of discrete systems – 11 hours.
4. Providing classroom training (processing lecture material and preparing reports on laboratory results) – 16 hours.
5. Providing individual tasks (performing an individual calculation task and completing it) – 12 hours.
6. Providing semester control (preparation for module control) – 10 hours.

Course materials and recommended reading

Main literature
1. Lee H.H. Finite Element Simulations with ANSYS Workbench 2021.-SDC Publications, 2021. (https://www.sdcpublications.com/Textbooks/Finite-Element-Simulations-ANSYS-Workbench/ISBN/978-1-63057-456-7/)
2. ANSYS 2023R1. Mechanical User's Guide. ANSYS Inc., Southpointe, 2600 Ansys Drive, Canonsburg, PA 15317, 2023. (https://ansyshelp.ansys.com/Views/ Secured/corp/v231/en/pdf/Workbench_Users_Guide.pdf)

Additional literature
1. Zienkiewicz O.C., Taylor R.L. and Zhu J.Z. The Finite Element Method: Its Basis and Fundamentals. Butterworth-Heinemann, Sixth edition, 2013. 802 p.
2. Ansys Student - Free Software Download. ANSYS Inc. 2023. https://www.ansys.com/academic/students

Assessment and grading

Criteria for assessment of student performance, and the final score structure

Content module 1 (LCs) – maximum 50 points: computer test (40 random short questions with 4 answer options, of which 1 is correct - 1.25 points for each correct answer) or exam (1 theoretically detailed question and a practical task to solve an engineering problem for modeling processes in a structure) - maximum 20 points for a correct answer to a question and a maximum of 20 points for a correctly solved and analyzed problem).
Content module 2 (LPCs) – maximum 50 points: 47 laboratory work tasks (maximum 1.0 points for each completed and submitted laboratory work task) and 1 individual calculation task (maximum 3.0 points for completed and defended calculation task).
Total – maximum 100 points.

Grading scale

Total points	National	ECTS
90–100	Excellent	A
82–89	Good	B
75–81	Good	C
64–74	Satisfactory	D
60–63	Satisfactory	E
35–59	Unsatisfactory (requires additional learning)	FX
1–34	Unsatisfactory (requires repetition of the course)	F

Norms of academic integrity and course policy

The student must adhere to the Code of Ethics of Academic Relations and Integrity of NTU "KhPI": to demonstrate discipline, good manners, kindness, honesty, and responsibility. Conflict situations should be openly discussed in academic groups with a lecturer, and if it is impossible to resolve the conflict, they should be brought to the attention of the Institute's management.
Regulatory and legal documents related to the implementation of the principles of academic integrity at NTU "KhPI" are available on the website: http://blogs.kpi.kharkov.ua/v2/nv/akademichna-dobrochesnist/

Approval

Approved by	Date August 30, 2023	Head of the department Oleksii VODKA
	Date August 30, 2023	Guarantor of the educational and professional program (1 year 4 months) Oleksiy LARIN