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
2

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

Anton Vasiliev
(assistant lecturer and laboratory practical teacher)
Anton.Vasiliev@khpi.edu.ua

Candidate of Technical Sciences (PhD), Senior Researcher, Assistant Professor.
Experience – 20+ years. Author of 100+ scientific papers.

Lecturer of the disciplines: "Computer Aided Design and Engineering", "Computer modeling of dynamics and vibration protection of rotary machines", "Modeling of nonlinear processes in CAE".

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 in linear and especially nonlinear formulations, namely structural strength and stability, natural and forced vibrations, random vibrations, quasi-static and transient dynamic processes without and taking into account structural nonlinearities, 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 natural and forced stationary and non-stationary vibrations, linear and nonlinear stability, static, quasi-static and dynamic strength, taking into account geometric and physical nonlinearities and contact interaction, including elements of geometric modeling, types of finite elements, meshing methods using various problem statements and corresponding them types of finite elements, construction of a calculation model with boundary conditions and loads of different types, output of calculation results, assessment of the quality of the finite element model and calculation results using various criteria.

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 with 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 for determining the characteristics of natural and forced stationary and non-stationary vibrations, linear and nonlinear stability, static, quasi-static and dynamic strength, taking into account geometric and physical nonlinearities of contact interaction 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 natural and forced stationary and non-stationary vibrations, linear and nonlinear stability, static, quasi-static and dynamic strength taking into account geometric and physical nonlinearities and contact interaction 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.

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;
PC10: Ability to develop mathematical methods and algorithms for computer modelling of nonlinear physical phenomena and processes in innovative technological systems;
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.
LO16. Be able to develop mathematical methods and algorithms for computer modelling of nonlinear physical phenomena and processes in innovative technological systems.
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 90 hours (3 ECTS credits): lectures - 32 hours, laboratory classes - 16 hours, self-study - 42 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;
PP 3. Modeling in CAE systems.

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

The course "Modeling in CAE systems" (2nd 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 and nonlinear physical processes in structures and their elements to assess the structural stability, static and dynamic strength of these objects with taking into account the geometric and physical nonlinearities and contact interaction..

Program of the course

Topics of the lectures

MODULE #1. Lecture classes (LCs) 1 credit / 16 hours “THEORETICAL BASIS FOR MODELING LINEAR AND NONLINEAR BEHAVIOR OF STRUCTURAL ELEMENTS IN CAE SYSTEM ANSYS WORKBENCH” (one class covers 1 topic in 2 hours)

Topic 1. Linear and nonlinear structural dynamics in ANSYS Workbench.

Introduction to structural dynamics in ANSYS Workbench:1. Definition and purpose. 2. Types of dynamic analysis. 3. General equation of motion. 4. Basic concepts and terminology.

Topic 2. Linear and nonlinear structural dynamics in ANSYS Workbench.

Damping in ANSYS Workbench: 1. Damping definition. 2. Types of damping. 3. General equation of motion. 4. Viscous damping in single-DOF vibration. 5. Damping matrices. 6. Numerical damping. 7. Damping summary.

Topic 3. Linear and nonlinear structural dynamics in ANSYS Workbench.

Modal analysis in ANSYS Workbench: 1. Definition and purpose. 2. Theory and terminology. 3. Eigenfrequencies and mode shapes. 4. Participation factors and effective mass. 5. Mode extraction methods – undamped. 6. Contact in modal analysis. 7. Analysis settings. 8. Damped modal analysis. 9. Mode extraction methods – damped. 10. Linear perturbation (prestress state). 11. Modal analysis based on linear perturbation.

Topic 4 Linear and nonlinear structural dynamics in ANSYS Workbench.

Harmonic analysis in the ANSYS Workbench: 1. Definition and purpose. 2. Theory and terminology. 3. Contact in harmonic analysis. 4. Damping in harmonic analysis. 5. Loads and boundary conditions. 6. Full harmonic analysis. 7. Mode-superposition harmonic analysis. 8. Taking into account linear perturbations in harmonic analysis when performed by both methods.

Topic 5 Linear and nonlinear structural dynamics in ANSYS Workbench.

Spectrum analysis in ANSYS Workbench: 1. Definition and purpose. 2. Generating the response spectrum. 3. Types of analyses. 4. Single point response spectrum analysis. 5. Mode combination methods. 6. Rigid response. 7. Missing mass response. 8. Multi-point response spectrum analysis. 9. Performing analysis and recommendations.

Topic 6. Linear and nonlinear structural dynamics in ANSYS Workbench.

Random vibration analysis in ANSYS Workbench: 1. Definition and purpose. 2. Power spectral density. 3. Theory and terminology. 4. PSD curve fitting. 5. Analysis settings. 6. Loads and boundary conditions. 7. Output and analysis of random vibrations characteristics.

Topic 7. Linear and nonlinear structural dynamics in ANSYS Workbench.

Transient analysis in ANSYS Workbench: 1. Definition and purpose. 2. Solution methods. 3. Nonlinearities. 4. The Newton-Raphson method.

Topic 8. Linear and nonlinear structural dynamics in ANSYS Workbench.

Transient analysis in ANSYS Workbench: 1. Full method of analysis and its settings. 2. Initial Conditions. 3. Loads and boundary conditions. 4. Analysis by mode-superposition method. 5. Taking into account the prestress state in the analysis by mode-superposition method.

Topic 9. Basic structural nonlinearities in ANSYS Workbench.

Overview and procedure for accounting of structural nonlinearities in ANSYS Workbench: 1. Nonlinear behavior. 2. Types of nonlinearities. 3. Nonlinear solution using linear solvers. 4. Nonlinear FEA issues. 5. Building a nonlinear model. 6. Obtaining a nonlinear solution.

Topic 10. Basic structural nonlinearities in ANSYS Workbench.

Restart and nonlinear controls in ANSYS Workbench: 1. Restart controls. 2. Newton-Raphson process. 3. Force and moment convergence. 4. Displacement and rotation convergence. 5. Summary of convergence criteria. 6. Line search procedure. 7. Solution stabilization. 8. Reviewing nonlinear results.

Topic 11. Basic structural nonlinearities in ANSYS Workbench.

Contact interaction in structures in ANSYS Workbench: 1. Introduction to contact. 2. Basic concepts. 3. Contact formulations. 4. Detection methods. 5. Trim contact. 6. Penetration tolerance. 7. Contact stiffness. 8. Pinball region. 9. Symmetric vs. asymmetric behavior. 10. Body types in contact. 11. Postprocessing contact results.

Topic 12. Basic structural nonlinearities in ANSYS Workbench.

Rate independent plasticity in ANSYS Workbench: 1. Background on elasticity and plasticity. 2. Yield criteria. 3. Flow rule. 4. Hardening rules. 5. Material data input. 6. Analysis settings. 7. Reviewing results. 8. Summary.

Topic 13. Basic structural nonlinearities in ANSYS Workbench.

Linear and nonlinear analysis of structural stability: 1. General information about structural stability. 2. Procedure for linear analysis of buckling (eigenvalue problem). 3. General information about methods of nonlinear analysis of buckling. 4. Nonlinear stabilization. 5. Summary of structural stability.

Topic 14. Basic structural nonlinearities in ANSYS Workbench.

Solver input and nonlinear solution monitoring in ANSYS Workbench: 1. Solver output. 2. Monitoring the solution. 3. Newton-Raphson residuals. 4. Example cases. 5. Tips on solving the model. 6. Summary.

Topic 15. Basic structural nonlinearities in ANSYS Workbench.

Mesh Nonlinear Adaptivity *in ANSYS Workbench*: 1. General information about rezoning. 2. Introduction to mesh nonlinear adaptivity. 3. Understanding the criteria (energy based, position based, contact based, mesh-quality based). 4. Procedure. 5. 3D nonlinear adaptivity example. 6. Nonlinear adaptivity limitations.

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 (linear and nonlinear dynamics of structures and taking into account the main types of structural nonlinearities and contact interaction in ANSYS Workbench).

Topics of the workshops

There are no classes.

Topics of the laboratory classes

MODULE #2. Laboratory practical classes (LPCs) 2 credits / 32 hours “COMPUTER MODELING OF LINEAR AND NONLINEAR BEHAVIOR OF STRUCTURAL ELEMENTS IN THE CAE SYSTEM ANSYS WORKBENCH” (one class covers 2 topics in 2 hours)

Topic 1. Linear and nonlinear structural dynamics in ANSYS Workbench.

Task 1 – Static strength and modal analyzes of a solid flywheel structure in ANSYS Workbench.

Topic 2. Linear and nonlinear structural dynamics in ANSYS Workbench.

Task 1 – Modal analysis of a single degree of freedom oscillator at different damping levels in ANSYS Workbench.

Topic 3. Linear and nonlinear structural dynamics in ANSYS Workbench.

Task 1 – Modal analysis of a plate with a hole in ANSYS Workbench.

Task 2 – Modal analysis of a statically loaded volumetric aircraft wing structure in ANSYS Workbench.

Topic 4. Linear and nonlinear structural dynamics in ANSYS Workbench.

Task 1 – Harmonic analysis of a solid flywheel structure in ANSYS Workbench.

Task 2 – Harmonic analysis of a single degree of freedom oscillator at different damping levels in ANSYS Workbench.

Task 3 – Modal and harmonic (under the action of two harmonic loads from rotating machines) analyzes of a beam fixed on both sides in ANSYS Workbench.

Topic 5. Linear and nonlinear structural dynamics in ANSYS Workbench.

Task 1 – Static, modal and spectral analyzes of a beam-shell suspension bridge model under static and seismic loads in ANSYS Workbench.
Topic 6. Linear and nonlinear structural dynamics in ANSYS Workbench.
Task 1 – Modal and random vibration analyzes of a beam-truss structure in ANSYS Workbench.
Task 2 – Modal and random vibration analyzes of a 3D tensile test specimen with fatigue evaluation in ANSYS Workbench.
Topic 7. Linear and nonlinear structural dynamics in ANSYS Workbench.
Task 1 – Dynamic analysis by two methods of the transient process in the spatial model of a portal crane when applying a remote force in ANSYS Workbench.
Topic 8. Linear and nonlinear structural dynamics in ANSYS Workbench.
Task 1 – Dynamic analysis of the transient process in a 3D model of installation for wiring and welding during rotating motion in ANSYS Workbench.
Topic 9. Basic structural nonlinearities in ANSYS Workbench.
Task 1 – Static strength comparative analysis of plate structure using small and large deflection theories under the same loads and boundary conditions in ANSYS Workbench.
Task 2 – Static strength comparative analysis of a shell structure using small and large deflection theories under the same loads and boundary conditions in ANSYS Workbench.
Topic 10. Basic structural nonlinearities in ANSYS Workbench.
Task 1 – Use of Restart Control technology to address the lack of a converged solution in ANSYS Workbench.
Task 2 – Using Line Search technology to improve solutions for large deviations in ANSYS Workbench.
Topic 11. Basic structural nonlinearities in ANSYS Workbench.
Task 1 – Static strength analysis of the contact interaction of two plate structures in an axisymmetric setting to determine contact pressure and penetration in ANSYS Workbench.
Task 2 – Static strength analysis of the contact interaction between the ball and the ball coupling in an axisymmetric formulation to determine the contact pressure in the ball joint in ANSYS Workbench.
Topic 12. Basic structural nonlinearities in ANSYS Workbench.
Task 1 – Quasi-static strength analysis of a disc spring in an axisymmetric formulation, taking into account the plastic behavior of the metal under sign-aligned load in ANSYS Workbench.
Topic 13. Basic structural nonlinearities in ANSYS Workbench.
Task 1 – Static strength and stability analyzes of a beam structure of a thin column in the linear formulation of ANSYS Workbench.
Task 2 – Analysing the post-buckling behavior of a thin column beam structure using nonlinear stabilization tools in ANSYS Workbench.
Task 3 – Comparative modal analyzes of a statically loaded volumetric structure taking into account linear and nonlinear contacts in ANSYS Workbench.
Topic 14. Basic structural nonlinearities in ANSYS Workbench.
Task 1 – Static strength analysis of a shell model of a spring structure under different contact interaction settings in ANSYS Workbench.
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 concepts, equations and methods of the theory of forced oscillations of discrete systems when solving problems of different classes under different types of dynamic load – 6 hours.
2. Basic concepts, equations and methods of linear and nonlinear theories of structural stability – 6 hours.
3. Basic concepts, equations and methods used in analyzing the behavior of discrete systems taking into account geometric, physical and contact nonlinearities - 6 hours.
4. Providing classroom training (processing lecture material and preparing reports on laboratory results) – 8 hours.
5. Providing individual tasks (performing an individual calculation task and completing it) – 8 hours.
6. Providing semester control (preparation for module control) – 8 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/) S. GRAHAM KELLY
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. Kelly G. Mechanical Vibrations: Theory and Applications, SI. Cengage Learning, Stamford, CT, USA, 2012 (http://160592857366.free.fr/joe/ebooks/Mechanical%20Engineering%20Books%20Collection/VIBRATIONS/mechVib%20theory%20and%20applications.pdf)
3. Yoo C.H., Lee S.C. Stability of Structures. Principles and Applications. Elsevier, Burlington, MA, USA, 2011. (http://parastesh.usc.ac.ir/files/1550995691902.pdf)
4. Chakrabarty J. Theory of Plasticity. Third edition., Elsevier Butterworth-Heinemann, Oxford. UK, 2006. (http://liet-clmc.yolasite.com/resources/theory%20of%20plasticity%20-%20chakrabarthy.pdf)
5. Szeptyński P. A Short Introduction to the Theory of Plasticity. Cracow University of Technology, 2020. (http://limba.wil.pk.edu.pl/kpmoc/images/stories/kpmoc/pracownicy/PSz/TSiP/TP-ang.pdf)
6. Popov V.L. Heß M., Willert E. Handbook of Contact Mechanics. Exact Solutions of Axisymmetric Contact Problems. Springer-Verlag GmbH Deutschland, 2018. (https://library.oapen.org/bitstream/id/2ea98fbf-d001-4dff-8a63-d3555f559b5b/1006864.pdf)
7. Fischer-Cripps A.C. Introduction to Contact Mechanics. Second Edition. Springer, 2007. (http://160592857366.free.fr/joe/ebooks/Automative%20engineering%20books/Introduction%20to%20Contact%20Mechanics.pdf)
8. 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: 23 laboratory work tasks (maximum 2.0 points for each completed and submitted laboratory work task) and 1 individual calculation task (maximum 4.0 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