KAZAKH engineering

ANSYS Course Catalog 2026

Deformable Solid Mechanics

kz-engineering.com

KAZAKH engineering

Basic Course.
Getting Started with ANSYS Mechanical

Duration — 2 days

This course serves as an alternative introduction to working with ANSYS Mechanical in the ANSYS Workbench environment. It contains no formal lectures; instead, every topic is explained through step-by-step demonstrations of setting up and solving a static structural analysis of a valve assembly. All practical exercises are performed on the same geometry—a conveyor shaft assembly—and represent consecutive stages of setting up a static strength analysis.

Unlike other introductory courses, this one does not cover modal or thermal analysis. Instead, it provides a more rigorous and structured methodology for conducting static structural simulations.

Therefore, the course is suitable not only for new users of ANSYS Mechanical, but also for engineers who are just beginning to work with finite element analysis (FEA) in general.

Course Outline:

- Introduction
- Problem definition
- Modeling approach
- Geometry, materials, and coordinate systems
- Connections
- Mesh
- Loads and supports
- Results and validation
- Approach to creating a more accurate model
- Geometry modification
- More realistic connections
- Methods for generating a more suitable mesh
- Additional loads and supports
- Advanced post-processing and validation
- Parameters and associativity

Standard Examples:

- Static structural analysis of a valve assembly (instructor demonstration)
- Static structural analysis of a conveyor shaft assembly with bearings

KAZAKH engineering

Basic Course.
Fundamentals of Nonlinear Analysis in ANSYS Mechanical

Duration — 2 days

The course is intended for users familiar with ANSYS Mechanical who want to enhance their proficiency by learning advanced nonlinear analysis techniques. It covers various material nonlinearities, contact modeling, and solution tools for nonlinear problems.

The course combines both theoretical and practical components. Topics include plasticity, nonlinear contacts, geometric nonlinearity, stabilization techniques, and sealing analysis.

Course Outline:

- Introduction
- Overview of nonlinearities
- Restart settings
- Fundamentals of nonlinear contact
- Metal plasticity
- Nonlinear stabilization
- Nonlinear diagnostics
- Adaptive remeshing

Standard Examples:

- Large displacements
- Use of restart analysis
- Contact stiffness
- Comparison of symmetric and asymmetric contact
- Multilinear isotropic hardening of metals
- Linear and nonlinear buckling
- Nonlinear solution diagnostics
- Nonlinear adaptive remeshing

KAZAKH engineering

Basic Course.
Introduction to ANSYS Aqwa

Duration — 3 days

The course is dedicated to the fundamentals of using ANSYS Aqwa and is intended for both experienced and beginner users. It covers the effects of ocean waves on ships, offshore platforms, and other structures, as well as related structural analyses.

Course Outline:

- Introduction to ANSYS Workbench
- Hydrodynamic diffraction
- Hydrodynamic response
- Connections in ANSYS Aqwa
- Analysis of multi-body systems
- Slender body hydrodynamics
- Transfer of loads from ANSYS Aqwa to ANSYS Mechanical
- Brief overview of classical Aqwa programs (Line/Librium/Fer/Naut/Drift)

Standard Examples:

- Hydrodynamic diffraction of a vessel
- Hydrodynamic response of a ship
- Modeling of FPSO platform connections
- Boat–pier interaction
- Flow around offshore platform structures
- Data transfer to ANSYS Mechanical for structural analysis of the platform

KAZAKH engineering

Basic Course.
Introduction to ANSYS Explicit Dynamics and AUTODYN – Part 1

Duration — 2 days

The course is dedicated to studying simulation techniques for dynamic processes using ANSYS Explicit Dynamics. It covers the Explicit Dynamics interface, material models, meshing strategies, and key features of the ANSYS AUTODYN solver. The practical part includes examples of solving various dynamic problems such as drop tests, impact between a projectile and a target, analysis of pre-stressed structures under dynamic loading, and more.

Course Outline:

- Problem setup in Explicit Dynamics
- Introduction to Workbench Explicit Dynamics
- Mesh generation
- Material models
- Connections
- Solver settings and parameterization

Standard Examples:

- Cylinder impact on a rigid target (Taylor test)
- Compression of an aluminum can
- Drop test of a circuit board
- Impact on a pre-stressed cylinder
- Drop test of a pre-stressed gas cylinder
- Comparison of different mesh types
- 2D simulation of projectile–target interaction
- 1D simulation of shock wave propagation
- Fan blade failure (blade-off event)
- Oblique impact between a projectile and a target
- Impact of a falling body on a reinforced concrete beam
- Penetration of a reinforced concrete barrier
- Use of Mass Scaling in the compression of an aluminum beam
- Drop test of a plastic container
- Dynamics of a pre-stressed beam
- What-if analysis for cylinder impact scenarios
- Compression of an aluminum can filled with liquid

KAZAKH engineering

Basic Course.
Introduction to ANSYS AUTODYN – Part 2

Duration — 2 days

The course covers the theoretical foundations of solving dynamic problems using explicit methods in ANSYS AUTODYN and is intended for users who have completed the “Introduction to Explicit Dynamics and AUTODYN” course.

It includes the use of Lagrangian, Eulerian, Arbitrary Lagrangian–Eulerian (ALE), and meshless (SPH) solvers, as well as their coupling.

The practical part of the course covers impact, explosion, and projectile–target interaction problems, among others.

Course Outline:

- Introduction to AUTODYN
- Multimaterial Euler solver
- AUTODYN interface
- AUTODYN fundamentals
- Material models
- Integration of AUTODYN with ANSYS
- Euler solver for explosion modeling
- Arbitrary Lagrangian–Eulerian (ALE) solver
- Mesh-free solver (SPH)
- Use of parallel computing in AUTODYN

Standard Examples:

- Compression of a filled aluminum can
- Drop test of a filled container
- Projectile–target interaction (2D)
- Simulation of a structure under impulsive loading
- Helmet impact analysis
- Simulation of shaped charge jet interaction with a target
- Simulation of shaped charge jet formation
- Explosion loading of a target
- Projectile–target interaction (2D) launched from ANSYS Workbench
- Explosion loading of a ship
- Mine detonation
- Urban explosion scenario
- Improvised explosive device (IED) detonation
- Bird strike on an aircraft wing (bird strike resistance)

KAZAKH engineering

Basic Course.
Introduction to ANSYS MAPDL

Duration — 3 days

The course is intended for new users or for those who use ANSYS Mechanical or ANSYS MAPDL occasionally and want to gain a solid foundation in working within the classic environment. The course combines lectures with practical problem-solving. It covers model preparation (preprocessing), solver setup, and result post-processing; provides an overview of mesh generation; and explains how to apply boundary conditions and loads using both the classic ANSYS MAPDL interface and APDL commands.

A brief overview of the interaction between the MAPDL environment and ANSYS Mechanical is also included.

Course Outline:

- Introductory demonstration
- Fundamentals of finite element theory
- APDL
- Geometry creation and import
- Selection logic
- Coordinate systems
- Element attributes
- Mesh generation
- Boundary conditions and loads
- Solvers
- Post-processing
- Modal and harmonic analysis using mode superposition
- Constraint equations
- Use of parameters
- 2D analysis
- Beam and shell elements
- Contact
- Bolt pretension
- Special load elements
- Coupled analysis
- Command objects in ANSYS Mechanical

Standard Examples:

- APDL
- Geometry creation
- Geometry import
- Selection logic
- Coordinate systems
- Element attributes
- Mesh generation
- Boundary conditions
- Modal analysis
- Harmonic analysis using mode superposition
- Periodic boundary conditions
- Moment transfer
- *GET functions for creating remote points
- Writing results to a text file
- Tabular loading
- Hanging bracket
- Pressure vessel
- Channel section
- Beams and shells
- Bonded contact
- Fastener and standard contact
- MPC contact using a pilot node
- Bolt pretension
- Load application using SURF154
- Convection modeling
- One-way thermomechanical coupling
- Thermomechanical coupling using special elements
- Pulsed thermoelectric heater simulation

KAZAKH engineering

Basic Course.
Introduction to ANSYS Mechanical

Duration — 3 days

The course is intended for new users or for those who use ANSYS Mechanical occasionally and wish to develop a solid foundation of core skills.

It combines theoretical lectures with practical problem-solving. The course covers model preparation (preprocessing), solver setup, and result post-processing; provides a brief overview of mesh generation in ANSYS Meshing; and explains how to apply boundary conditions and loads.

Course Outline:

- Introduction
- Fundamentals and interface of ANSYS Mechanical
- Preprocessing
- Mesh generation
- Contacts, joints, beams, and springs
- Remote boundary conditions
- Static structural analysis
- Modal analysis
- Steady-state thermal analysis
- Multi-step analysis
- Results processing and post-processing
- CAD import and parameterization
- Submodeling method (additional chapter)
- Linear buckling analysis (additional chapter)
- Beam modeling (additional chapter)
- Shell modeling (additional chapter)

Standard Examples:

- ANSYS Mechanical fundamentals
- 2D gear interaction
- Creating named selections
- Object generator
- Mesh generation on an assembly example
- Contact management
- Use of joints
- Application of remote boundary conditions
- Constraint equations
- Linear structural analysis of a pump assembly
- Creating connections using beam elements
- Natural frequency analysis of a metal frame
- Steady-state thermal analysis of a pump cover
- Multi-step analysis
- Mesh quality evaluation
- Project parameter management
- Linear buckling analysis (additional example)
- Application of the submodeling method (additional example)
- Beam modeling (additional example)
- Shell modeling (additional example)
- Shell submodeling (additional example)

KAZAKH engineering

Basic Course.
Heat Transfer Simulation in ANSYS Mechanical

Duration — 1 day

The course is dedicated to modeling heat conduction in solids and surface radiative heat transfer (with convective heat flux applied as a boundary condition) using ANSYS Mechanical. It covers element types, material properties, boundary conditions, solver settings, and post-processing tools. Both steady-state and transient analyses are addressed, including problems with phase change. The course also includes examples of using command snippets written in APDL.

Course Outline:

- Introduction
- Theoretical foundations of heat conduction
- Working in the preprocessor
- Boundary conditions and solver settings
- Steady-state heat conduction analysis
- Nonlinear heat conduction problems
- Transient heat conduction analysis
- Special topics: phase change heat transfer and use of APDL command snippets
- Thermo-mechanical stress analysis

Standard Examples:

- Heat conduction in a rod
- Heat transfer in a heating coil
- Thermal contact
- Heat conduction with surface radiation
- Heat transfer in a solenoid
- Heat transfer in a finned wall with temperature-dependent thermal conductivity and heat transfer coefficients
- Transient heat transfer with cyclically varying volumetric heat generation
- Heat transfer during solidification of an aluminum roller
- Coupled thermo-structural analysis

KAZAKH engineering

Basic Course.
Introduction to ANSYS Motion

Duration — 2 days

The course is intended for new users or for those who use ANSYS Motion occasionally and want to develop a solid foundation of core skills.

It combines theoretical lectures with practical problem-solving. The course covers model preparation (preprocessing), solver setup, and result post-processing, as well as a brief overview of the Car, Links, and Drivetrain toolkits, and the ANSYS EasyFlex solver.

Course Outline:

- Introduction
- Fundamentals and interface of ANSYS Motion
- Model structure
- Problem setup (preprocessing)
- Joints and contacts
- Modal and harmonic analysis
- Dynamics of mechanical systems
- Template-based modeling
- Car, Links, and Drivetrain toolkits
- Applications and capabilities of the ANSYS EasyFlex solver

Standard Examples:

- Dynamic analysis of a crank-slider mechanism
- Harmonic analysis of a brushless motor
- Fatigue analysis of a suspension arm
- NVH analysis of a drivetrain
- Timing belt drive analysis
- Vehicle dynamics (half-car and full-car models)
- Printer drop test

Note:

The basic course can be extended with user-specific case studies upon request.

KAZAKH engineering

Specialized Course. ANSYS nCode

Duration — 2 days

This course is intended for users familiar with ANSYS Mechanical.

It covers the theoretical foundations of fatigue analysis under proportional and non-proportional loading conditions. The course includes stress-life (S–N) and strain-life (E–N) approaches, definition of load histories, fatigue analysis under vibration loading, and practical examples of structural durability assessment using these methods.

Course Outline:

- Introduction to fatigue analysis
- Integration of ANSYS Workbench and nCode DesignLife
- Graphical interface of nCode DesignLife
- Import of FEA results
- Material properties
- Load time history definition
- Load blocks
- Stress-life fatigue analysis (S–N)
- Strain-life fatigue analysis (E–N)
- Vibration fatigue analysis
- Standalone use of nCode DesignLife

Standard Examples:

- Ready-to-use project
- Simple high-cycle fatigue with constant amplitude
- Import of FEA models
- Assignment of material properties
- Definition of load sequences
- Import of load history from FEA results
- Load blocks
- Stress-life fatigue analysis (S–N)
- Strain-life fatigue analysis (E–N)
- Elastic–plastic correction

KAZAKH engineering

Specialized Course.
Fatigue Analysis in ANSYS Fatigue

Duration — 1 day

The course is intended for users familiar with the basics of ANSYS Mechanical who want to enhance their skills by learning fatigue analysis of structures. The Fatigue module enables durability assessment under simple cyclic loading conditions.

The course combines both theoretical background and practical applications.

Course Outline:

- Fundamentals of fatigue phenomena
- Stress-life fatigue: constant amplitude, proportional loading
- Stress-life fatigue: variable amplitude, proportional loading
- Stress-life fatigue: constant amplitude, non-proportional loading
- Strain-life fatigue: constant amplitude, proportional loading
- Frequency-domain (vibration) fatigue

Standard Examples:

- Introduction — stress-life method (S–N)
- Variable amplitude, proportional loading — stress-life approach
- Constant amplitude, non-proportional loading — stress-life approach
- Strain-life approach (E–N)
- Frequency-domain fatigue

KAZAKH engineering

Specialized Course.
Introduction to ANSYS Additive Suite

Duration — 2 days

The course explores the capabilities of the ANSYS Additive Suite, including Additive Prep, Additive Print, and Workbench Additive, for simulating additive manufacturing processes.

The program covers the functionality of Additive Prep for model preparation, design considerations, and the workflow for setting up simulations in ANSYS Workbench and Additive Print, as well as the key features of these tools.

The course is intended for users familiar with the basics of ANSYS Mechanical.

Course Outline:

Additive Prep

- Introduction to ANSYS Additive Prep

Additive Print

- Introduction to the DMLS (Direct Metal Laser Sintering) process
- Introduction to ANSYS Additive Print
- Visualization software ParaView
- Calibration and validation
- Results evaluation

ANSYS Workbench

- General overview of additive manufacturing simulation
- Design for additive manufacturing (DfAM)
- Simulation workflow in ANSYS Workbench Mechanical
- Use of APDL commands for additive manufacturing simulation

Standard Examples:

Additive Prep

- Working in ANSYS Additive Prep

Additive Print

- Analysis of a rectangular beam in ANSYS Additive Print
- Post-processing of the rectangular beam and support optimization
- Calibration process setup
- Evaluation of results for a cylindrical rod
- Assessment of orientation effects

ANSYS Workbench

- Simulation of the additive manufacturing process in ANSYS Workbench Mechanical
- Support structure generation

KAZAKH engineering

Specialized Course.
Introduction to ANSYS Composite PrepPost

Duration — 2 days

The course covers both theoretical and practical aspects of modeling composite structures using ANSYS Composite PrepPost.

It includes the process of creating finite element models of composite structures, draping analysis tools, ply orientation definition, and post-processing techniques such as layer-by-layer failure criteria evaluation, delamination analysis, and local buckling assessment. The course also provides a detailed overview of the integration of ANSYS Workbench into the Composite PrepPost workflow.

Course Outline:

- Fundamentals of composite materials
- Introduction to ANSYS Composite PrepPost
- Overview of the typical modeling and analysis workflow in ANSYS Composite PrepPost
- Local coordinate systems (rosettes)
- Oriented element sets
- Rule sets for element selection
- Draping simulation in ANSYS Composite PrepPost
- Modeling composites using solid finite elements
- Failure criteria analysis for composite materials
- Parameters in ACP

Standard Examples:

- Modeling of a sandwich panel
- Definition of ply layup for a T-joint
- Use of rule sets
- Modeling of a sandwich panel
- Modeling composites using solid finite elements
- Kitesurf board
- Working with parameters

KAZAKH engineering

Specialized Course.
Application of Beam and Shell Models in ANSYS Mechanical

Duration — 1 day

The course provides a detailed overview of the features, capabilities, and tools for using beam and shell elements in ANSYS Mechanical. In addition to the elements themselves, it also covers tools for connecting bodies at the mesh level, which is the most common type of connection in models composed of beams and shells.

The course is intended for users already familiar with the interface of ANSYS Workbench Mechanical.

Course Outline:

- Modeling using beam elements
- Modeling using shell elements
- Creating mesh-level connections

Standard Examples:

- Analysis of a beam model of a floating platform
- Analysis of a shell model of a pressure vessel
- Analysis of a submodel of the vessel (continuation of the previous example)
- Working with a T-joint
- “Bonding” a shell model at the mesh level using a barge structure example

KAZAKH engineering

Specialized Course.
Introduction to ANSYS Workbench LS-DYNA

Duration — 2 days

The course covers the theoretical foundations of setting up, solving, and post-processing dynamic problems in ANSYS LS-DYNA within the ANSYS Workbench Mechanical environment. It includes topics such as integration of ANSYS LS-DYNA into Workbench and provides materials on solving problems using the Lagrangian formulation.

The practical part includes impact problems, projectile–target interaction, dynamic buckling, and other dynamic analyses.

Course Outline:

- Theoretical foundations of explicit dynamics and Workbench LS-DYNA
- Solver settings, boundary conditions, and working with rigid bodies
- Post-processing using Workbench LS-DYNA and LS-PrePost
- Modeling of connections
- Quasi-static analysis
- Material models and Engineering Data
- Mesh generation
- Element formulations
- LS-DYNA keyword (card) input

Standard Examples:

- Taylor impact test
- Rotary draw bending
- Drop test wizard in Workbench LS-DYNA
- Post-processing in LS-PrePost
- Pipe impact analysis
- Quasi-static analysis
- Mesh generation
- Drop test
- Aircraft wing bird strike simulation

KAZAKH engineering

Specialized Course.
Rigid Body Dynamics in ANSYS

Duration — 1 day

The course covers modeling of systems composed of purely rigid bodies, as well as systems that include both rigid and deformable bodies. It also provides a detailed overview of joint definitions and their capabilities.

The course is intended for users familiar with the basics of ANSYS Mechanical.

Course Outline:

- Introduction to multibody system analysis
- Simulation of rigid body dynamics
- Joints
- Analysis of systems with rigid and deformable bodies

Standard Examples:

- Assembly creation
- Drive mechanism
- Crank–slider mechanism

KAZAKH engineering

Specialized Course. Dynamics in ANSYS

Duration — 2 days

The course includes theoretical foundations of the equations of motion and their application in various dynamic analyses. It is intended for users familiar with the basics of ANSYS Mechanical.

The practical part covers modal, harmonic, spectral, random vibration, and transient dynamic analyses.

Course Outline:

- Introduction to dynamics
- Damping
- Modal analysis
- Cyclic symmetry
- Prestress effects
- Harmonic analysis
- Spectral analysis
- Random vibration analysis
- Transient dynamic analysis

Standard Examples:

- Vibration analysis of a flywheel
- Study of damping effects
- Free vibration of a plate with a hole
- Bladed disk
- Cyclic symmetry of a bevel gear
- Linear perturbation of two beams
- Harmonic response of a clamped plate
- Spectral analysis of a prestressed suspension bridge
- Response of a metal frame to an acceleration spectrum
- Impact simulation between a wheel and a metal block
- Transient analysis of a crane assembly
- Shaft rotation in transient analysis

KAZAKH engineering

Specialized Course.
Using MAPDL Commands in ANSYS Workbench

Duration — 2 days

The course covers the use of command objects to extend the functionality of ANSYS Workbench.

It explains the fundamental principles of command-based modeling and the structure of the classic ANSYS MAPDL environment, as well as how to use command snippets within ANSYS Mechanical.

The course is intended for users familiar with the basics of ANSYS Mechanical.

Course Outline:

- Introduction
- Introduction to APDL
- Attributes
- Post-processing
- APDL commands
- Using APDL in ANSYS Workbench Mechanical

Standard Examples:

- Introductory problem in ANSYS MAPDL
- Selection logic
- Ventilation duct
- APDL script
- Forces in spot welds
- Spot weld parameters
- Parameter arrays
- Reinforcement elements
- Forces in spot welds in ANSYS Mechanical

KAZAKH engineering

Specialized Course.
Use of Nonlinear Contacts in ANSYS

Duration — 2 days

The course is intended for users familiar with linear and nonlinear analysis in ANSYS Mechanical who want to improve their skills in working with nonlinear contacts.

It covers contact modeling techniques, the use of APDL commands, bolt pretension, and gasket modeling.

Course Outline:

- Introduction
- Overview of contact technology
- Surface setup
- Using APDL commands for contact definition
- Bolt pretension modeling
- Gasket modeling
- General contact

Standard Examples:

- Automatic contact detection
- Using Worksheet for contact setup
- Contact surface configuration
- Contact stabilization
- Frictional contacts
- Fluid pressure loading
- Bolt pretension
- Bolt pretension with large rotation
- Maximum shear stress
- Wear modeling
- Bolt pretension modeling
- Gasket modeling

KAZAKH engineering

Specialized Course.
Use of Nonlinear Material Models in ANSYS

Duration — 1 day

The course includes theoretical foundations of nonlinear material behavior, covering both basic and advanced material models, as well as approximation of experimental curves. It is intended for users familiar with linear and nonlinear analysis in ANSYS Mechanical.

The practical part covers the Chaboche model, as well as plasticity, hyperelasticity, and viscoelasticity models.

Course Outline:

- Introduction
- Plasticity
- Element technology
- Viscoplasticity
- Creep
- Hyperelasticity
- Viscoelasticity
- Advanced material models

Standard Examples:

- Chaboche model
- Creep
- Hyperelasticity
- Viscoelasticity

Additional Topics:

- Hill’s anisotropic plasticity model
- Gray cast iron plasticity model
- Microplane model for concrete
- Shape memory alloy models

KAZAKH engineering

Specialized Course.
Fracture Mechanics in ANSYS Mechanical

Duration — 1 day

The course covers the theoretical foundations of setup, solution, and post-processing of fracture mechanics problems. It includes methods for evaluating stress intensity factors, J-integral, and other key fracture mechanics parameters using various crack modeling techniques.

Course Outline:

- Introduction to fracture mechanics
- Crack modeling tools
- Modeling of a semi-elliptical crack
- Arbitrary-shaped crack
- Geometry-based crack modeling
- Virtual Crack Closure Technique (VCCT) and delamination modeling
- SMART crack growth method
- Overview of XFEM crack growth modeling

Standard Examples:

- Semi-elliptical crack
- Arbitrary-shaped crack
- Predefined crack
- Virtual Crack Closure Technique (VCCT)
- Debonding in materials
- Crack growth simulation using the SMART method

KAZAKH engineering

Specialized Course.
Fundamentals of ALE and SPH Simulations in LS-DYNA

Duration — 2 days

The course covers the theoretical foundations of solving dynamic problems using explicit formulation in ANSYS LS-DYNA and is intended for users who have completed the “Introduction to ANSYS LS-DYNA” course.

It includes the main approaches to setting up and solving problems using Euler, ALE, and SPH formulations, as well as coupling techniques between these formulations and Lagrangian elements.

The practical part includes impact, explosion, and projectile–target interaction problems, among others.

Course Outline:

- Fundamentals of the ALE method
- Interaction of bodies and materials
- Domain creation
- Explosion modeling
- Fundamentals of the SPH method
- S-ALE method

Standard Examples:

- Taylor test using ALE formulation
- Penetration using Euler formulation
- Leakage control
- Penetration by a Lagrangian projectile
- Bird strike simulation
- Cylinder impact
- Use of shell container
- Shaped charge simulation
- Hypervelocity impact using SPH
- Sloshing
- S-ALE method in Workbench LS-DYNA
- S-ALE method in LS-PrePost
- Application of the Load Blast Enhanced card

KAZAKH engineering

Specialized Course.
Best Practices and Efficient Workflows in ANSYS Mechanical

Duration — 2 days

The course is intended for users already familiar with ANSYS.

It presents best practices and techniques developed from practical experience with ANSYS and user support, aimed at simplifying workflows and achieving accurate results. The course also covers the fundamentals of the finite element method and numerical integration techniques used in ANSYS Mechanical for solving solid mechanics problems.

Special attention is given to answering key practical questions such as: “How can the model size be reduced without losing accuracy?”, “What type of finite element mesh should be used?”, and “How can we ensure that the obtained solution is sufficiently accurate?”

Course Outline:

- Overview of FEM: simple and complex problems
- Element theory: basic equations and numerical integration
- Types of elements
- Model preparation
- Use of symmetry in modeling
- Loads and boundary conditions
- Solution and verification of results

Standard Examples:

- Mesh convergence study
- Integration options
- Element selection
- Comparison of results using different modeling approaches
- Stress singularities
- Application of symmetry
- Solution and post-processing

KAZAKH engineering

Specialized Course.
Rotor Dynamics in ANSYS

Duration — 1 day

The course is intended for users familiar with the basics of ANSYS Mechanical and who have completed the “Dynamics” module.

It includes theoretical background on the dynamics of rotating bodies and practical material for solving rotor dynamics problems such as modal analysis, Campbell diagram generation, stability analysis and determination of critical speeds, harmonic analysis for evaluating vibration amplitudes of a rotating rotor under imbalance, as well as transient analysis for simulating rotor response during startup, shutdown, and external dynamic excitations.

Course Outline:

- Introduction to rotor dynamics
- Modal analysis
- Harmonic analysis
- Types of finite elements with support for Coriolis matrix and gyroscopic effects

Standard Examples:

- Nelson rotor
- Cantilever rotor
- Critical speed map (Campbell diagram)
- Harmonic response
- General axisymmetric elements

KAZAKH engineering

Specialized Course.
Creating and Configuring ACT Extensions

Duration — 2 days

The course is dedicated to developing custom ACT extensions to extend the functionality of ANSYS Mechanical.

During the course, participants learn Python programming and receive step-by-step instructions for creating various types of extensions.

The course is intended for advanced users of ANSYS Mechanical.

Course Outline:

- Fundamentals of ACT
- Basics of Python programming
- Debugging scripts using the IronPython Console

Standard Examples:

- Installation of a ready-made ACT extension
- Creation and installation of a binary extension
- Exploring the IronPython Console
- Development of a custom extension for user-defined loads
- Development of a custom extension for user-defined results
- Development of a custom extension using APDL commands

KAZAKH engineering

Specialized Course.
Topology Optimization in ANSYS Mechanical

Duration — 2 days

The course is intended for users of ANSYS Mechanical who want to develop core skills in solving topology optimization problems.

It combines theoretical lectures with practical problem-solving. The course covers the general solution workflow, optimization problem setup, objective functions, constraints, and post-processing of topology optimization results in ANSYS SpaceClaim.

Course Outline:

- Material distribution along load paths
- Topology optimization based on static analysis
- Working with CAD
- Geometry reconstruction
- Optimization of a helicopter blade component
- Use of manufacturing constraints
- Topology optimization based on modal analysis
- Applications of topology optimization

Standard Examples:

- Michell structure
- STL file export
- Topology optimization based on static analysis
- Multiple load cases
- Working with CAD
- Geometry reconstruction
- Topology optimization based on modal analysis
- Spatial structure optimization

KAZAKH engineering

Specialized Course.
Advanced Capabilities of ANSYS Mechanical

Duration — 3 days

This course is dedicated to advanced features of working in ANSYS Mechanical and also includes the specialized course “Using MAPDL Commands in ANSYS Workbench.”

It covers topics such as advanced post-processing, import and export of various data formats, as well as the fundamental principles of command usage, the structure of the classical ANSYS MAPDL environment, and modeling using command snippets in ANSYS Workbench Mechanical.

The course is intended for users familiar with the basics of ANSYS Mechanical.

Course Outline:

- Advanced post-processing
- Data import using External Data
- Model import and assembly creation
- Solution workflow
- Introduction to APDL
- Attributes
- Post-processing
- APDL commands
- Using APDL in Workbench Mechanical

Standard Examples:

- Post-processing of an axisymmetric pressure vessel model according to ASME standards
- Stress export/import using a beam bending example
- Importing an aircraft fuselage model into Mechanical from a CDB file
- Introductory problem in MAPDL
- Selection logic using a primitive example
- Working with MAPDL using a ventilation duct example
- Creating an APDL script using a simple cantilever beam problem
- Evaluating forces in spot welds using MAPDL
- Creating force parameters in spot welds and exporting them to an external file
- Macro for saving stress and strain fields

About us

- **KazakhEngineering** is a certified official partner of **ANSYS** in the Republic of Kazakhstan.
- We implement advanced digital engineering technologies, develop and adapt solutions tailored to the specific needs of each enterprise, enhancing the efficiency of simulation, modeling, and technical decision-making.
- We also provide specialist training and comprehensive support at every stage of using engineering software.

Ansys part of SYNOPSYS®

KAZAKH engineering

Контакты

ТОО «КазахИнжиниринг»
Алматы, ул. Гоголя, 73
+7 (778) 372-01-52
reception@kz-engineering.com