KAZAKH engineering

ANSYS Course Catalog 2026

kz-engineering.com

Table of contents

KAZAKH engineering

Deformable Solid Mechanics

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

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

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

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

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)

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

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)

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

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.

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

Electromechanics

Basic Course.
Electromagnetic Field Simulation in ANSYS Maxwell 2D/3D

Duration — 4 days

The course is dedicated to electromagnetic field simulation in 2D planar, axisymmetric, and full 3D formulations using ANSYS Maxwell. It covers steady-state, harmonic, and transient electromagnetic analyses. Participants will learn how to evaluate key characteristics such as magnetic field intensity, magnetic flux density, magnetic flux, inductance matrices, and other parameters. The course also addresses material properties, boundary conditions, solver settings, and post-processing tools.

For users with no prior experience in ANSYS Twin Builder (formerly Simplorer) or ANSYS Maxwell, part of the course is devoted to learning the interface, as well as creating geometric and mesh models.

The course is recommended for beginner users.

Course Outline:

- Theoretical foundations
- Working with the graphical user interface
- Types of analysis (at this stage, participants choose a preferred focus area)
- Material properties and working with material libraries
- Types of boundary conditions and modeling simplification methods
- Mesh generator and mesh operations
- Adaptive solution and estimation of computational errors
- Calculation of capacitance and inductance
- Demagnetization of nonlinear permanent magnets and determination of operating points based on magnetization
- Working with the postprocessor
- ANSYS Optimetrics module: parametric studies and user-defined variables
- Transient analysis: problems involving motion of model components
- Direct and indirect methods for evaluating losses in electrical steel under alternating magnetic fields
- Managing electromechanical models using the ANSYS Maxwell Circuit Editor
- Introduction to system-level simulation in ANSYS Simplorer (Twin Builder)
- Basic optimization tasks
- Simple examples of coupled problems

Basic Course.
Electric Machine Simulation in ANSYS Motor-CAD

Duration — 3–9 days (depending on the selected course content)

The course is dedicated to the simulation of electric machines using a multiphysics analysis module that includes electromagnetic, thermal, and mechanical solvers. It covers several types of electric machines with detailed explanations of solver settings.

The course is recommended for beginner users. Upon completion, participants receive guidance for independent work and supporting materials.

Course Outline:

- Permanent Magnet Synchronous Machine (PMSM)
- Direct-On-Line (DOL) induction machine
- Inverter-driven induction machine
- Synchronous reluctance machine
- System-level modeling with ANSYS Motor-CAD
- Multiphysics optimization of electric machines using ANSYS optiSLang
- Thermal analysis in ANSYS Motor-CAD

Basic Course.
Electric Machine Simulation in ANSYS Maxwell 2D/3D

Duration — 4 days

The course is dedicated to electromagnetic field simulation in 2D planar, axisymmetric, and full 3D formulations, with a focus on electric machine applications, using ANSYS Maxwell. It covers steady-state, harmonic, and transient analyses, including problems involving motion.

Participants will learn to evaluate key characteristics such as magnetic field intensity, flux density, magnetic flux, as well as inductance and capacitance matrices, among others. The course also addresses material properties, boundary conditions, solver settings, and post-processing tools.

For users with no prior experience in ANSYS Twin Builder (formerly Simplorer) or ANSYS Maxwell, part of the course is devoted to learning the interface, as well as creating and simplifying geometric and mesh models.

The duration may vary significantly depending on the participants' preferences. This course complements the ANSYS Maxwell 2D/3D course and is more specifically focused on electric machines. User-specific problem-based courses are not included in this module.

The course is recommended for beginner users. Upon completion, participants receive guidance for independent work and supporting materials.

Course Outline:

- Specialized solution for electric machines: ANSYS RMxprt
- Selection of electric machine type
- Working with tabular input forms: defining key geometric dimensions, material properties, winding parameters, and more
- Analytical calculation of electric machine characteristics
- Parametric studies: defining user variables, performing parametric analysis, parallel computing, and running simulations on remote computing resources
- Working with the postprocessor
- Examples of model performance optimization
- Examples of creating 2D/3D field simulations in ANSYS Maxwell based on an RMxprt model
- Setup of motion-related simulations
- Use of built-in macros for creating rotating machine models
- Modeling power and control circuits in ANSYS Twin Builder (formerly Simplorer) in combination with analytical RMxprt models or finite element models in ANSYS Maxwell 2D/3D

Specialized User-Oriented Course. Electromagnetic Field Simulation in ANSYS Maxwell 2D/3D

Duration — depends on the complexity of the task

Completion of the basic course is a mandatory prerequisite.

A technical specification is prepared, and time is allocated for its development prior to the training.

The course is dedicated to electromagnetic field simulation in 2D planar, axisymmetric, and full 3D formulations using ANSYS Maxwell. It covers steady-state, harmonic, and transient analyses, including problems involving motion.

Participants will learn to evaluate key characteristics such as magnetic field intensity, magnetic flux, inductance matrices, and other parameters. The course also addresses material properties, boundary conditions, solver settings, and post-processing tools.

The course includes solving transient problems with moving components and is recommended for users already familiar with simulation methodologies.

Course Outline:

Based on the provided simulation models, problems of magnetostatics, harmonic fields, and transient processes are solved. Electrostatic problems are considered separately.

Specialized User-Oriented Course. Electromagnetic Field Simulation in ANSYS Maxwell 2D/3D. Multiphysics Analysis

Duration — depends on the complexity of the task

The course requires knowledge at the level of basic courses in ANSYS Maxwell 2D/3D. For multiphysics applications, additional knowledge of ANSYS IcePak, ANSYS Fluent, ANSYS Meshing, and Fluent Meshing is recommended.

A technical specification is prepared in advance, and time is allocated for course development.

The course focuses on electromagnetic field simulation in 2D planar, axisymmetric, and full 3D formulations. It includes steady-state, harmonic, and transient analyses, as well as evaluation of key characteristics such as magnetic field intensity, magnetic flux, inductance and capacitance matrices, and the thermal state of the model.

The course also covers material properties, boundary conditions, solver settings, and post-processing tools. It includes transient simulations with moving components and is recommended for users familiar with simulation methodologies.

Course Outline:

Based on the provided simulation models, problems of magnetostatics, harmonic field analysis, and transient processes are solved, as well as multiphysics problems involving electromagnetic–thermal coupling.

Specialized Course. Thermal Analysis of Electric Machines in ANSYS Fluent and ANSYS Maxwell. Multiphysics Simulations

Duration – 4 days

The course requires knowledge at the level of a basic course in electric machine simulation using ANSYS Maxwell 2D/3D and conjugate heat transfer modeling in ANSYS Fluent.

The course demonstrates the workflow for solving conjugate heat transfer problems in electric machines using ANSYS Fluent, with input data generated in ANSYS Maxwell. It includes transient electromagnetic simulations with motion, post-processing of results, preparation of the geometry in ANSYS SpaceClaim, and generation of a high-quality mesh for thermal analysis using ANSYS Fluent Meshing.

By agreement, the user's own electric machine model can be used during the course.

The course is recommended for users familiar with simulation methodologies.

Course Outline:

- Setting up the problem in ANSYS Maxwell
- Preparing the geometry in ANSYS SpaceClaim
- Creating the mesh in ANSYS Fluent Meshing
- Solver setup in ANSYS Fluent and coupling with ANSYS Maxwell
- One-way and two-way electromagnetic–thermal coupling simulations

Specialized Course.
Electromagnet Simulation in ANSYS Maxwell

Duration – 4 days

The course requires knowledge at the level of a basic course in electromagnetic field simulation using ANSYS Maxwell 2D/3D.

It is dedicated to modeling electromagnetic fields of electromagnets in axisymmetric and full 3D formulations. The course covers both steady-state and transient magnetic analyses, including problems involving motion, with the use of post-processing tools for result evaluation.

Topics also include material properties, boundary conditions, solver settings, and post-processing tools.

The course is recommended for users familiar with simulation methodologies.

Course Outline:

- General approaches to electromagnet modeling
- Magnetic problems with moving armature
- Methods for defining loads and springs for the armature
- Eddy current effects in solid ferromagnetic materials
- Lateral forces acting on the armature
- Automation of electromagnet coil definition
- Residual magnetization of electrical steel
- Optimization of geometric dimensions

Specialized Course.
Induction Heating Simulation Using ANSYS System Coupling

Duration – 3 days

The course requires knowledge at the level of a basic course in electromagnetic field simulation using ANSYS Maxwell 2D/3D, as well as thermal analysis using ANSYS Mechanical (Transient) or ANSYS Fluent.

The course focuses on simulating alternating electromagnetic fields in a 3D formulation and their effect on heating ferromagnetic materials. It includes solving harmonic magnetic problems, determining induced currents, and calculating volumetric heat generation in the inductor and the ferromagnetic workpiece. The ANSYS System Coupling environment is used to couple the electromagnetic solver with a transient thermal solver, enabling accurate time-dependent analysis of the heating process under strong variations in material properties.

The course is recommended for users familiar with simulation methodologies.

Course Outline:

- Harmonic magnetic analysis in ANSYS Maxwell
- Temperature-dependent material properties
- Thermal model setup in ANSYS Transient Thermal
- Setting up synchronization variables for time-dependent changes in current, frequency, and inductor position
- Configuration of ANSYS System Coupling for multiphysics simulations

Specialized Course.
Electric Machine Simulation in ANSYS Maxwell 2D/3D

Duration – 5 days

The course requires knowledge at the level of a basic course in electric machine simulation using ANSYS Maxwell 2D/3D.

It focuses on electromagnetic field simulation of electric machines in 2D and 3D formulations. The course covers steady-state and transient magnetic analyses, along with post-processing techniques for evaluating results. It also addresses material properties, boundary conditions, solver settings, and post-processing tools. Transient simulations with motion are included.

In the extended version of the course, the ANSYS optiSLang tool is used for optimization, along with ACT-based custom applications for generating efficiency maps of electric machines.

The course is recommended for users familiar with simulation methodologies.

Course Outline:

- Building geometric models of electric machines using the UDP (User-Defined Primitives) library
- Evaluation of cogging torque effects on torque quality
- Mesh operations for discretization of simulation models
- Power balance in an electric machine
- Optimization of the magnetic system using ANSYS optiSLang
- Creation of a reduced-order model (ROM) of a permanent magnet synchronous machine
- Demagnetization of permanent magnets
- Fast convergence to steady-state for induction motors
- Electric Machine Toolkit tools for generating efficiency maps

High-Frequency Devices

KAZAKH engineering

Basic Course. Introduction to ANSYS HFSS

Duration — 4 days

The course covers the process of setting up a project in ANSYS HFSS for the analysis of basic antennas and microwave (RF) devices.

Participants will become familiar with the graphical user interface of ANSYS Electronics Desktop (AEDT), where the HFSS (High Frequency Structure Simulator) tool is integrated. The course walks through the full HFSS workflow, including geometry creation, definition of boundaries and simulation domains, wave and lumped ports, solution setup, frequency sweeps, and post-processing of results.

Post-processing includes plotting S-parameters and visualizing electromagnetic fields on geometry. The course is practice-oriented, with approximately 60% of the time dedicated to hands-on exercises and 40% to theoretical concepts.

In addition, the course introduces high-performance computing (HPC), parametric studies, and optimization (Optimetrics) in ANSYS HFSS.

Course Outline:

- Boundary conditions and simulation domain
- Project setup: meshing and frequency sweep
- Post-processing: S-parameters and field visualization on geometry
- Geometry creation in ANSYS HFSS
- Wave ports and lumped ports
- High-performance computing (HPC) and optimization analysis (Optimetrics)

Basic Course.
PCB Analysis in ANSYS HFSS 3D Layout

Duration — 4 days

This is a basic course on printed circuit board (PCB) analysis using ANSYS HFSS 3D Layout within ANSYS Electronics Desktop (AEDT). The course is intended for users who are just starting to work with this software.

It covers the user interface, PCB layer visualization, layout handling, ports, vias (interlayer connections), definition of simulation boundaries, and more. Practical sessions include analysis of a differential via transition, a spiral inductor, a planar antenna array, a PCB section, and a mobile phone example.

Course Outline:

- Working with models in ANSYS HFSS 3D Layout and data visualization
- Solvers, mesh generation, and solution setup
- Types of ports in ANSYS HFSS 3D Layout
- Simulation domain boundaries
- Eye diagram extraction and time-domain reflectometry (TDR)
- Preparing a mobile phone model for analysis

Basic Course.
Fundamentals of ANSYS SIwave

Duration — 4 days

ANSYS SIwave is an advanced tool for the analysis and design of complex printed circuit boards (PCBs). It enables the extraction of S-parameters and RLCG characteristics. ANSYS SIwave supports a wide range of analyses, including impedance scanning, DC IR drop analysis, and time-domain reflectometry (TDR).

Course Outline:

- Setting up Signal Integrity (SI) analysis, including impedance scanning, SYZ, and TDR
- Setting up Power Integrity (PI) analysis, including SYZ and RLGC
- Configuration of PCB components (R, L, C), ports, and terminations
- Setup of SPICE models and S-parameter components
- DC IR drop analysis of a PCB

KAZAKH engineering

Specialized Course.
Working with 3D Components, Boundaries, Ports, and Mesh in ANSYS HFSS

Duration — 2 days

The course covers full 3D modeling of high-frequency structures using the finite element method (FEM) in ANSYS HFSS. It includes hierarchical placement of 3D components in different coordinate systems within a single HFSS project.

Topics also include working with boundary conditions for 2D surfaces and 3D absorbing boundary conditions for volumetric structures. Special attention is given to wave port setup (including port sizing, 2D field distributions, operating modes, etc.).

In addition, the course covers the operation of the FEM solver, adaptive mesh refinement, meshing strategies for multi-resonant structures, mesh filling, and solution setup.

Course Outline:

- Working with 3D components in ANSYS HFSS
- Boundary conditions
- Port setup in ANSYS HFSS: operation and application of different port types for various problems
- Features of mesh generation using the FEM method

Specialized Course.
Antenna Analysis in ANSYS HFSS

Duration — 4 days

The course focuses on working in ANSYS HFSS through antenna analysis examples. It covers boundary conditions (Absorbing, Radiation PML/ABC, Radiation, FE-BI), as well as near-field and far-field calculations. The course also includes dynamic linking with the Circuit Design environment and optimization using Optimetrics.

It addresses modeling using the finite element (FE) solver, which generates volumetric meshes based on the finite element method.

Additionally, the course introduces Boundary Integral (BI) methods for creating 2D surface meshes and analyzing large electromagnetic structures, including IE regions, Physical Optics (PO), and the SBR+ method.

The final sections cover hybrid FE-BI approaches (Finite Element – Boundary Integral).

Course Outline:

- Near-field and far-field analysis
- Excitations and various types of polarization
- Boundary conditions for antenna analysis
- Dynamic linking with the circuit editor (matching problem example)
- Solving optimization problems
- Integral Equation (IE) region
- Antenna analysis using hybrid regions and boundary integral methods (IE, PO, SBR+)
- Solving problems using hybrid regions

Specialized Course.
ANSYS HFSS SBR+: Antenna Placement Analysis on Platforms

Duration — 2 days

The course focuses on the SBR+ method in ANSYS HFSS. It covers antenna placement on electrically large platforms and the calculation of antenna coupling.

Practical sessions include analysis of an antenna mounted in a vehicle side mirror and the influence of the car body on antenna performance. The course also examines coupling between a mobile antenna inside the vehicle and an antenna in the rear-view mirror, as well as coupling between an in-cabin Wi-Fi antenna and an external antenna located in a garage, taking into account the material properties of the garage and road surface.

Course Outline:

- Use of 3D components
- Antenna placement on electrically large platforms
- Antenna coupling analysis
- Use of SBR+ for coupling analysis between antennas in a vehicle and in a garage

Specialized Course.
Analysis of Layered Structures in ANSYS HFSS 3D Layout

Duration — 3 days

The course covers the analysis of layered structures using ANSYS HFSS 3D Layout within the ANSYS Electronics Desktop (AEDT) environment.

It includes integration of 3D components and PCB elements in HFSS 3D-Layout, as well as optimization and analysis of passive and active circuits.

Course Outline:

- 3D components in ANSYS HFSS 3D Layout
- Optimization of via (interlayer transition) structures
- Modeling and analysis of a microstrip filter
- Dynamic linking between ANSYS HFSS 3D Layout and the Circuit editor
- ECAD Xplorer: preprocessing large GDSII files before importing into ANSYS HFSS 3D Layout

Specialized Course.
PCB Analysis in ANSYS HFSS 3D Layout

Duration — 3 days

The course covers high-speed PCB analysis using ANSYS HFSS 3D Layout. It includes working with Padstack to define PCB layer structures and creating differential via transitions with ground planes across multiple layers.

It also demonstrates integration of a package model, PCB, and IBIS model within a single environment using the Nexxim transient simulator in ANSYS Electronics Desktop (AEDT).

The course includes configuring a PCB model for eye diagram analysis and covers device optimization techniques.

In addition, participants will set up connectors, packages, and PCB models within one environment, select appropriate solvers, and perform DC IR drop analysis.

Course Outline:

- ANSYS HFSS 3D Layout: Working with the Padstack tool for PCB layer configuration
- PCB assembly process and integration of IBIS controllers
- PCB setup for eye diagram analysis
- Connector analysis on a PCB
- DC IR drop analysis of a PCB

Specialized Course.
EM Analysis in ANSYS SIwave

Duration — 3 days

The course covers the analysis of electromagnetic compatibility (EMC) issues in printed circuit boards (PCBs) and methods for mitigating them using the EM analysis capabilities of ANSYS SIwave.

It includes an example of analyzing a memory interface system for compliance with DDR4 electrical standards. The course also covers power integrity analysis of packages and PCBs using the SIwave-PSI solver.

Course Outline:

- Electromagnetic interference (EMI)
- EMI scanner
- Induced voltage and resonant modes
- Near-field and far-field extraction
- DDR4 channel setup and analysis
- SIwave PSI: full-wave solution for PCB power integrity analysis

KAZAKH engineering

Fluid Dynamics and Heat Transfer

Basic Course.
Computational Fluid Dynamics (CFD) in ANSYS CFX

Duration — 3 days

The course is aimed at developing basic skills in using ANSYS CFX. It combines theoretical lectures with practical problem-solving sessions. The course covers the structure and use of the preprocessor, solver manager, and postprocessor; mesh import; definition of the computational domain and physical models; boundary and initial conditions; mesh interfaces; the CEL and CCL languages; transient processes; porous media; additional variables; source terms; and output file handling.

Course Outline:

- Introduction to CFD methodology. Introduction to ANSYS Workbench
- Creation of the computational domain, boundary conditions, and source terms
- Analysis of results using ANSYS CFD-Post
- Solver setup and analysis of output files
- Mesh interfaces and moving domains
- Heat transfer modeling
- Turbulent flow modeling
- Transient simulation
- Practical recommendations for CFD modeling
- CFX Expression Language (CEL) and CFX Command Language (CCL)
- Appendix: Using macros written in Perl to automate CFX projects

Standard Examples:

- Flow with heat transfer in a mixing T-shaped channel
- Multicomponent flow and post-processing
- Transonic flow around a NACA0012 airfoil
- Axial fan stage
- Cooling of a processor by natural convection and radiation
- Simulation of a Kármán vortex street

Basic Course.
Computational Fluid Dynamics (CFD) in ANSYS Fluent

Duration — 3 days

The course is designed for both beginners with no prior experience in ANSYS Fluent and users who already have some experience and wish to systematize their knowledge.

The main objective of the course is to teach the fundamentals of working in the ANSYS Fluent environment, develop practical skills in solving computational fluid dynamics (CFD) problems, and build a solid foundation in numerical simulation of fluid and gas flows.

Course Outline:

- Introduction to CFD methodology. Overview of the graphical interface of ANSYS Fluent and the main stages of project setup
- Mesh zones and boundary conditions
- Analysis of simulation results
- Solver settings
- Turbulence modeling
- Heat transfer modeling
- Practical recommendations for CFD modeling
- Transient flow simulation
- Appendix: Advanced physical models — moving zones and dynamic mesh models
- Appendix: Advanced physical models — multiphase flows

Standard Examples:

- Flow simulation in a manifold
- Mixing elbow
- Mixing T-junction: influence of solver settings on results (case study)
- Post-processing of results using a tube bundle example
- Turbulent flow over a backward-facing step
- Cooling of an electronic board with natural convection and radiation
- Kármán vortex street

Basic Course.
Thermal Simulation of Electronic Devices in ANSYS Icepak

Duration — 3 days

The course is designed for engineers involved in the design of electronic systems. It covers all stages of performing three-dimensional numerical analysis of airflow distribution within a device, taking into account heat transfer processes such as conduction, convection, and radiation.

Course Outline:

- Introduction
- Interface structure and main steps of model creation
- ANSYS Icepak objects — air and solid regions
- Conformal mesh generation
- Solver settings
- Post-processing in ANSYS Icepak and ANSYS CFD-Post
- ANSYS Icepak objects — encapsulation (potting) regions, heat sinks, chips
- Non-conformal mesh generation
- Physical aspects of heat transfer processes and transient flow modeling
- Model parameterization
- Introduction to ANSYS Workbench and ANSYS DesignModeler
- Transfer of MCAD models to ANSYS Icepak using ANSYS DesignModeler
- Mesh generation (introduction, global settings, unstructured hexahedral mesh, hex-dominant mesh)
- Practical recommendations
- Parameterization and optimization using ANSYS DesignXplorer

Standard Examples:

- Creating a geometric model using ANSYS Icepak objects
- Generating a conformal mesh
- Solver setup, running the simulation, and results analysis
- Building a geometric model with ECAD import and use of ANSYS Icepak objects
- Generating a non-conformal mesh for a model with ECAD geometry
- Performing transient simulations
- Model parameterization
- Converting MCAD geometry to ANSYS Icepak format using ANSYS DesignModeler
- Creating a multi-level mesh model
- Optimization using ANSYS DesignXplorer

Basic Course.
Fundamentals of Simulation in ANSYS FENSAP-ICE

Duration — 4 days

The course is dedicated to studying the fundamentals of modeling droplet impingement and ice accretion under flight conditions using the specialized ANSYS FENSAP-ICE package.

It covers the structure of the software by modules: FENSAP — aerodynamic calculations, DROP3D — droplet impingement simulation, ICE3D — ice accretion simulation, and C3D/CHT3D — conjugate heat transfer modeling.

The course also considers the use of CFD packages such as ANSYS Fluent and ANSYS CFX for aerodynamic analysis as an alternative to FENSAP.

The course includes theoretical foundations of the methods used in the software and practical guidance on their application.

Course Outline:

- Introduction to FENSAP-ICE. Icing simulation system under flight conditions
- Aircraft icing in flight
- Fundamentals of theory
- User interface
- Aerodynamic solver
- Flow simulation module
- DROP3D module: Droplet impingement analysis. Some modeling tips
- DROP3D module: Supercooled Large Droplets (SLD)
- DROP3D module: Snow and ice crystals
- DROP3D: User interface
- ICE3D: Ice accretion simulation module
- ICE3D: User interface of the ice accretion module
- CHT3D: Conjugate heat transfer modeling (theory)
- C3D: Transient heat transfer module
- CHT3D: Conjugate heat transfer module
- Using ANSYS Fluent for aerodynamic analysis
- FENSAP-ICE user guide

Standard Examples:

- Introduction to the FENSAP-ICE interface
- Aerodynamic analysis of a NACA 0012 airfoil for smooth and rough surfaces
- Droplet impingement simulation on a NACA 0012 airfoil
- Ice accretion on a NACA 0012 airfoil
- Conjugate heat transfer simulation of a nacelle anti-icing system
- Using ANSYS Fluent for aerodynamic analysis and transferring results to FENSAP-ICE

Specialized Course. Multiphase Flows in ANSYS CFX

Duration — 2 days

The course covers methods for modeling multiphase flows (gas–liquid, solid particles–liquid or gas), including phenomena involving heat and mass transfer between phases. These methods are essential for solving problems such as cavitation, evaporation, boiling, condensation, and chemical reactions at phase interfaces.

The course requires knowledge at the level of a basic course in ANSYS CFX.

Course Outline:

- Introduction to multiphase flows
- Approaches to multiphase flow modeling
- Interphase momentum and heat transfer
- Free-surface flow modeling
- Multiphase modeling using the Lagrangian approach
- Multiphase modeling using the extended Lagrangian approach
- Interphase mass transfer
- Overview of MUSIG and DQMOM models
- Granular models in ANSYS CFX
- Phase change in multiphase multicomponent flows
- Practical recommendations for multiphase flow modeling in ANSYS CFX

Standard Examples:

- Flow in a bubble column
- Flow in a bubble column with additional effects
- Free-surface flow with surface tension
- Application of the algebraic slip model
- Droplet evaporation and Lagrangian particle model
- Rectangular bubble column with non-drag forces and MUSIG
- Wall boiling model
- Cavitation around a hydrofoil
- Simulation of sudden pipe depressurization
- Interphase mass transfer in multicomponent liquids

Specialized Course.
Multiphase Flows in ANSYS Fluent

Duration — 2 days

The course is dedicated to modeling multiphase flows using ANSYS Fluent. It covers a wide range of topics, including Lagrangian and Eulerian approaches, free-surface flows, dispersed phase modeling (motion of bubbles, droplets, and solid particles), granular flows, as well as interphase heat and mass transfer.

Course Outline:

- General aspects of multiphase flow modeling
- Volume of Fluid (VOF) method
- Discrete Phase Model (DPM) and Discrete Element Method (DEM)
- Eulerian multiphase model for gas–liquid flows
- Eulerian multiphase model for granular flows
- Mixture model

Standard Examples:

- Tank filling and draining (VOF)
- Discrete Phase Model (DPM)
- Bubble column simulation (Eulerian multiphase model)
- Simulation of uniform fluidization in a fluidized bed (Eulerian granular model)
- Bubble rise in a suspension (VOF Eulerian model)

Specialized Course.
Acoustic Simulation in ANSYS Fluent

Duration — 1 day

The course is aimed at providing a general understanding of aeroacoustic simulation, covering the main CFD approaches used to solve such problems and the specifics of their application.

It includes practical recommendations on mesh generation, turbulence modeling, and solver settings for aeroacoustic analysis, with particular emphasis on post-processing and interpretation of simulation results.

Course Outline:

- Introduction
- Computational Aeroacoustics (CAA)
- Acoustic analogy modeling
- Propeller noise modeling (Gutin's model)
- Broadband noise modeling
- Post-processing of acoustic simulation results

Standard Examples:

- Near-field noise simulation using direct aeroacoustic modeling
- Far-field noise simulation using the acoustic analogy method
- Gutin propeller noise model
- Broadband noise

Specialized Course.
Gradient-Based Optimization Using the Adjoint Solver in ANSYS Fluent

Duration — 2 days

The course covers the methodology of using the gradient-based optimization tool, the Adjoint Solver, integrated in ANSYS Fluent, to improve a target performance parameter.

The lecture materials provide detailed information on the tools used for optimization studies, result post-processing, and mesh deformation techniques. The practical part of the course demonstrates the full workflow of setting up and running an optimization case, including the impact of selected settings on the final results.

The course requires knowledge at the level of a basic course in ANSYS Fluent.

Course Outline:

- Introductory lecture: key definitions and overview of the Adjoint Solver workflow
- Objective functions (target parameters)
- Solver settings: discretization methods and solution stabilization
- Post-processing of Adjoint results
- Optimization tools: overview of mesh deformation methods, smoothing settings, and node movement
- Additional constraints for mesh deformation
- Automatic Gradient-Based Optimizer
- Creating a CAD model based on the optimized mesh

Standard Examples:

- Optimization of a U-bend
- Optimization of a NACA 0012 airfoil
- Influence of solver settings on convergence (Ahmed body case study)
- Post-processing of results (manifold example)
- Influence of different optimization tool settings on mesh deformation (S-bend example)
- Use of the automatic optimizer for multiple objectives and design points

Specialized Course.
Combustion Modeling in ANSYS Fluent

Duration — 2 days

The course covers combustion models for premixed, partially premixed, and non-premixed flows.

It also addresses topics such as chemical kinetics modeling, interaction between turbulent fluctuations and chemical reactions, liquid fuel spray modeling, combustion of solid fuel particles, and surface chemical reactions.

The course requires knowledge at the level of a basic course in ANSYS Fluent.

Course Outline:

- Introduction to reactive flow modeling
- Models for transport of chemical species
- Non-premixed combustion
- Premixed and partially premixed combustion
- Discrete phase modeling
- Surface reactions and pollutant formation
- Useful features and techniques for combustion modeling
- Radiative heat transfer

Standard Examples:

- Transport of species and combustion of gaseous fuel
- Application of the non-premixed combustion model
- Two-dimensional combustion chamber simulation
- BERL 300 kW case using the Magnussen and Laminar Flamelet models
- Premixed combustion in a conical chamber using the finite-rate chemistry model
- Simulation of Sandia Flame D using the Probability Density Function (PDF) model
- Modeling liquid-phase reactions in a closed impinging jet reactor using the transient Laminar Flamelet model
- Complex reactions in solid particle combustion
- Modeling heterogeneous reactions in granular flow using the Eulerian approach
- Evaporation of liquid droplets in a circular channel
- NOx formation during combustion with selective non-catalytic reduction (SNCR)
- Combustion simulation in a liquid rocket engine chamber using a real gas model
- Simulation of partially premixed combustion using LES and the Thickened Flame model

Specialized Course. Turbomachinery Simulation in ANSYS CFX

Duration — 1 day

The course is dedicated to the analysis of turbomachinery flow paths using ANSYS CFX.

It covers topics such as the use of rotating reference frames, interfaces between stationary and rotating domains, transient simulations, as well as post-processing techniques specific to turbomachinery applications.

Course Outline:

- Introductory lecture
- Theoretical foundations. Formulation of equations in rotating reference frames
- Single rotating reference frame
- Frozen Rotor model
- Mixing Plane model
- Sliding Mesh model
- Post-processing of flow path simulation results

Standard Examples:

- Simulation of flow between rotating disks using a single rotating reference frame
- Simulation of a blower using the Frozen Rotor model
- Simulation of an axial machine flow path using the Mixing Plane approach
- Simulation of an axial machine flow path using the Sliding Mesh method
- Working with turbomachinery flow simulation results
- Simulation of a centrifugal pump using a single rotating reference frame
- Simulation of a wind turbine using the Frozen Rotor and Sliding Mesh models
- Application of non-reflecting boundary conditions for transonic flow over a blade

Specialized Course.
Heat Transfer Simulation in ANSYS Fluent

Duration — 2 days

The course is dedicated to heat transfer modeling using ANSYS Fluent. The lecture materials include extensive theoretical background and provide a detailed overview of modeling the main heat transfer mechanisms—conduction, convection, and radiation.

Special attention is given to the application of turbulence models for heat transfer in boundary layers. In addition, the course covers the methodology for analyzing recuperative heat exchangers using the Dual-Cell approach.

Course Outline:

- Introduction to heat transfer theory
- Conduction
- Conjugate heat transfer
- Forced convection
- Natural convection
- Radiative heat transfer
- Solar radiation (insolation)
- Heat exchanger modeling
- Heat transfer in porous media

Standard Examples:

- Introductory example: flow with conjugate heat transfer through a heating coil
- Simulation of radiation and natural convection
- Heat transfer simulation in an automotive headlamp using the Discrete Ordinates and Monte Carlo models
- Turbulent flow with heat transfer in a compact heat exchanger
- Simulation of heat transfer between a flow and metal foam

Specialized Course.
Turbulence Modeling in ANSYS CFX or ANSYS Fluent

Duration — 1 day

The course is dedicated to the study of turbulence models implemented in ANSYS CFX and ANSYS Fluent, including eddy viscosity models, Reynolds stress models, wall function approaches, transition models, and scale-resolving models.

In the practical part of the course, participants solve several benchmark problems.

The course requires knowledge at the level of a basic course in ANSYS CFX or ANSYS Fluent.

Course Outline:

- Overview of engineering turbulence models
- RANS turbulence models in ANSYS CFD
- Eddy viscosity models (Zero Equation, k-ε, k-ω, BSL, SST)
- Reynolds stress models (LRR, SSG)
- Scalable wall functions
- Automatic wall function switching method
- Additional turbulence models
- Large Eddy Simulation (LES)
- Detached Eddy Simulation (DES)
- Transition model (laminar-to-turbulent transition)
- Scale-Adaptive Simulation (SAS)

Standard Examples:

- Flow over a flat plate with zero pressure gradient
- Separated flow in a diffuser
- Impinging jet
- Transitional flow around an airfoil

Specialized Course.
Chemical Reaction Modeling in ANSYS CHEMKIN

Duration — 2 days

The course is dedicated to modeling detailed chemical reaction mechanisms, including their development and reduction. It also covers surface heterogeneous reactions, ion-exchange processes, and the mathematical description of plasma.

Course Outline:

- Introduction
- Software interface and general concepts
- Ignition of fuel mixtures
- Partially stirred reactor networks
- Theoretical foundations of surface reactions
- Practical application of the software for surface chemistry problems
- Ion-exchange reactions
- Plasma
- Optimization of chemical mechanisms in Reaction Workbench

Standard Examples:

- Calculation of equilibrium time for nitric oxide (NO) formation reactions
- Ignition delay time calculation
- Laminar flame speed calculation
- Diffusion flame simulation
- Reactor network modeling for combustion in a turbine
- Catalytic oxidation of CH_4 on a platinum catalyst
- Modeling of an exhaust aftertreatment system using a reactor network
- Simulation of aluminum oxide deposition
- Optimization of a plasma reactor
- Reduction of a detailed chemical mechanism in Reaction Workbench

KAZAKH engineering

Specialized Course.
Application of Dynamic Meshes in ANSYS Fluent

Duration — 2 days

The course presents the capabilities of dynamic mesh techniques implemented in ANSYS Fluent. It focuses on methods such as remeshing, smoothing, and layering. The course also covers the use of User-Defined Functions (UDFs) to describe mesh motion, coupled simulations with the 6DOF solver, and other advanced features.

Course Outline:

- Overview of dynamic mesh methods
- Types of dynamic zones
- Layering (mesh layering)
- Spring-based mesh smoothing
- Local remeshing
- Coupled simulation with the 6DOF solver (six degrees of freedom)
- Use of User-Defined Functions (UDFs) for dynamic mesh modeling
- Additional features

Standard Examples:

- Mesh layering on simple geometries in 2D and 3D setups
- 2D simulation of oscillations of a metal plate and an internal combustion engine (ICE) combustion chamber using UDFs and the spring-based smoothing model
- Simulation of a gear pump using dynamic mesh with remeshing in a 2.5D setup and the CutCell method
- Gerotor pump simulation
- Vane pump simulation

Specialized Course.
Application of EWF and LWF Wall Film Models in ANSYS Fluent

Duration — 1 day

The course provides an in-depth study of the capabilities of the Eulerian Wall Film (EWF) and Lagrangian Wall Film (LWF) models for simulating the formation, flow, and detachment of liquid films on walls in ANSYS Fluent.

The EWF and LWF approaches significantly reduce the number of near-wall cells required compared to resolving the film using the Volume of Fluid (VOF) method.

Knowledge of EWF and LWF models is useful for solving problems such as wall condensation and evaporation in gas turbine chambers, heat exchangers, glazing, gas–liquid separation, annular flow in pipes, and surface coating processes.

To take this course, prior completion of the “Specialized Course: Multiphase Flows in ANSYS Fluent” is recommended.

Course Outline:

- Liquid droplet capture
- Liquid film transport
- Droplet formation on a liquid film
- Heat transfer between the film, gas, and wall
- Evaporation and condensation in the near-wall region
- Transitions between EWF, VOF, and DPM models
- Application of the Lagrangian Wall Film (LWF) model

Standard Examples:

- Formation and detachment of a water film over a backward-facing step
- Formation and detachment of a water film over an airfoil
- Evaporation of a fuel film from a tray
- Condensation of humid air in a heat exchanger
- Condensation of a gaseous phase on a thermosiphon wall using User-Defined Functions (UDFs)
- Prevention of cockpit window fogging
- Simulation of a mist eliminator using transitions between EWF, VOF, and DPM models
- Spray coating simulation using the LWF model

Specialized Course.
Application of User-Defined Functions (UDF) in ANSYS Fluent

Duration — 2 days

The course covers the use of User-Defined Functions (UDFs) written in C to extend the functionality of ANSYS Fluent.

These functions can be applied to a wide range of tasks—from defining custom source terms and boundary conditions to implementing user-specific physical models.

The course includes the fundamentals of C programming at a level sufficient for practical use, as well as detailed coverage of Fluent’s internal data structures and the interaction between UDFs and the main solver.

Course Outline:

- Introduction. Basics of programming, syntax, and data types
- Compilation and interpretation of user-defined functions
- Use of DEFINE macros
- Use of user-defined variables
- User-defined functions for parallel computing
- Using Workbench parameters together with user-defined functions
- User-defined functions for multiphase flows
- User-defined functions for the discrete phase model

Standard Examples:

- Defining a temperature profile as a boundary condition
- Defining an additional energy source in a cell zone
- User-defined memory (UDM): storing and processing arbitrary variables
- Writing data to a text file
- Using UDFs in parallel computations
- Flow in a channel with a porous obstacle
- Flow in a channel with a sinusoidal wall temperature distribution
- Using a custom temperature-dependent viscosity model
- Modeling transport of a user-defined scalar variable
- User-defined functions for modifying constants in empirical particle drag laws
- Simulation of sedimentation in a clarifier using user-defined functions
- Controlling dynamic mesh using user-defined functions

Geometry and Finite Element Meshing

Basic Course.
Mesh Generation in ANSYS Meshing

Duration — 1 day

The course is aimed at mastering the core meshing tools of the ANSYS Meshing software system.It covers various mesh generation methods and includes both theoretical materials and step-by-step practical examples.

Course Outline:

- Introduction to ANSYS Meshing
- Mesh generation methods
- Global mesh settings
- Local mesh settings
- Mesh quality assessment

Standard Examples:

- ANSYS Meshing Fundamentals
- ANSYS Meshing Methods
- Global Mesh Settings
- Local Mesh Settings
- Conical Combustion Chamber
- Pressure Vessel
- Explicit Dynamics: Projectile

Basic Course.
Mesh Generation in ANSYS TurboGrid

Duration — 1-2 days

The course is aimed at developing skills in creating mesh models of turbomachinery flow paths (turbines, fans, bladed pumps, and compressors) using the ANSYS TurboGrid software module.

Course Outline:

- Introduction to ANSYS TurboGrid
- Basic concepts
- User interface and workflow in the software
- Computational domain geometry
- Computational domain topology
- Mesh generation
- Automatic Topology and Mesh (ATM) method
- Mesh analysis and optimization

Standard Examples:

- Axial turbine rotor
- Axial compressor stage
- Splitter blade
- Axial fan
- Damaged blade
- Mixed-flow pump impeller

Basic Course.
Mesh Generation in ANSYS ICEM CFD

Duration — 2 days

The course is aimed at mastering the core meshing tools of the ANSYS ICEM CFD system. It covers topics such as importing and editing geometry, exporting mesh models to various solver formats, as well as creating and editing structured hexahedral meshes.

The course is intended for a wide range of users working with mesh models in fluid dynamics, structural analysis, heat transfer, and electromagnetics.

Course Outline:

- Introduction to ANSYS ICEM CFD
- Overview of capabilities, workflow, and toolset
- Topology strategies and block-structured mesh generation

Standard Examples:

- 2D pipe junction
- 3D elbow with a through hole
- 3D pipe junction with different diameters
- Mesh quality assessment and improvement using topology-based methods
- Application of block-structured meshing to complex geometry (crankshaft example)
- Coupling structured and unstructured meshes
- Working with block editing tools (structured mesh for a lug/eyelet example)
- Channel with a blade
- Hemisphere with a cubic cutout
- Surface mesh of a vehicle body

Basic Course.
ANSYS BladeModeler

Duration — 2 days

The course is aimed at developing skills in creating geometric models of turbomachinery impellers (turbines, fans, bladed pumps, and compressors) using the ANSYS BladeModeler software module.

Course Outline:

- Introduction
- ANSYS BladeModeler interface
- Overview of ANSYS turbomachinery software products
- BladeGen module
- BladeEditor option
- BladeEditor: Importing BGD (BladeGen Data)
- BladeEditor: Model creation

Standard Examples:

- Axial turbine rotor
- Low-pressure-ratio compressor impeller
- Axial fan blade
- Data Import Wizard
- Data transfer from CAD to BladeEditor, then to ANSYS TurboGrid
- Fan model creation and simulation
- Axial fan blade
- Radial turbine rotor using ANSYS BladeModeler
- Geometry and mesh generation for a centrifugal compressor impeller

Basic Course.
ANSYS DesignModeler

Duration — 2 days

The course is designed to teach the principles of creating, simplifying, and repairing 3D and 2D geometry using the ANSYS DesignModeler application. The software is built on the Parasolid kernel, uses a history-based modeling approach, and is fully integrated into the ANSYS Workbench environment.

Modeling can be performed using 2D sketches followed by feature-based operations, as well as with geometric primitives. In addition, the application supports topological parameterization and the creation of cross-sections for beam elements, which are subsequently used in ANSYS Mechanical.

Course Outline:

- Introduction
- Graphical User Interface (GUI)
- Planes and sketching mode
- Creation of 3D and 2D geometry
- Geometry simplification and repair
- Beam and shell modeling
- Working with imported geometry from CAD systems
- Parametric modeling

Standard Examples:

- ANSYS DesignModeler Fundamentals
- Working with sketches and creating landing gear geometry
- Working with primitives and creating 3D geometry of a muffler
- Simplification and repair of pump geometry
- Using beams and shells to create frame structures
- Model topology parameterization

Basic Course.
ANSYS SpaceClaim

Duration — 1 day

ANSYS SpaceClaim is designed for users who are not professional CAD specialists. The module enables the creation and editing of 3D geometric models and allows full parameterization of externally imported geometry.

The application is based on a direct modeling approach, meaning it does not rely on feature history. This simplifies working with large parameterized assemblies and enables rapid creation of the desired geometry.

In addition, the application supports the creation of cross-sections for beam elements, which are subsequently used in ANSYS Mechanical, including extraction from solid geometry.

Course Outline:

- Introduction and graphical user interface
- Working with 3D geometry
- Advanced geometry editing techniques
- Geometry simplification and repair
- Extracting mid-surfaces for shells and creating beam elements
- Defining material properties and using parameters

Standard Examples:

- Using sketches and the Pull tool
- Splitting imported geometry into separate components
- Refining geometry, creating fillets and chamfers
- Using operations to create solid geometry from surfaces
- Assembling individual parts into a structure
- Creating dynamic copies of objects
- Geometry simplification and removal of fillets
- Repairing imported geometry
- Using beams and shells
- Transferring models from SpaceClaim to Workbench

Basic Course.
Mesh Generation in ANSYS Fluent Meshing. Watertight Geometry Workflow (User Template)

Duration — 1 day

The course is dedicated to the fundamentals of mesh generation using the Watertight Geometry workflow in ANSYS Fluent Meshing. This workflow is primarily intended for meshing well-prepared, “clean” geometries of computational domains.

Using this workflow helps accelerate mesh generation thanks to its intuitive structure and parallel processing capabilities, even for new users.

The course includes both theoretical and practical sessions.

Course Outline:

- Introduction to Fluent Meshing
- Watertight Geometry Workflow: Template overview
- Watertight Geometry Workflow: Geometry import and surface mesh generation
- Watertight Geometry Workflow: Geometry description
- Watertight Geometry Workflow: Volume mesh generation

Standard Examples:

- Introduction to the Fluent Meshing interface. Watertight Geometry Workflow
- Mesh generation for a static mixer
- Mesh generation for a mixing tank
- Working with shared topology
- Defining rotational periodicity
- Mesh generation for aerodynamic analysis of an aircraft

Specialized Course.
Mesh Generation in ANSYS Fluent Meshing. Fault-Tolerant Workflow (User Template)

Duration — 1 day

The course is dedicated to the fundamentals of mesh generation using the Fault-Tolerant workflow in ANSYS Fluent Meshing. This workflow is primarily intended for meshing unprepared or “dirty” geometries of computational domains. It enables the straightforward use of advanced wrapping tools for geometry cleanup and mesh generation.

The course includes both theoretical and practical sessions.

Course Outline:

- Introduction to Fluent Meshing
- Fault-Tolerant Workflow: Template overview
- Fault-Tolerant Workflow: Wrapping tool
- Fault-Tolerant Workflow: Hole capping

Standard Examples:

- Introduction to the Fluent Meshing interface. Fault-Tolerant Workflow
- Mesh generation for a manifold
- Mesh generation around an airfoil
- Practice in controlling size functions
- Practice in hole capping
- Practice in using auxiliary surfaces

KAZAKH engineering

Multiphysics Simulations and Optimization

kz-engineering.com

Basic Course.
Introduction to ANSYS DesignXplorer

Duration — 1 day

The course is designed to familiarize users with methods for solving optimization problems and conducting other parametric studies using ANSYS DesignXplorer within the Workbench environment. The course covers all DX-implemented methods, including design of experiments (DOE), parameter correlation analysis, response surface modeling, optimization, and sensitivity analysis of the model to variations in input parameters. Basic experience with structural or CFD ANSYS packages is recommended.

Course Outline:

- Introduction
- Parameter correlation
- Design of experiments (DOE)
- Response surfaces
- Optimization methods
- Six Sigma analysis

Standard Examples:

- Introductory example
- Parameter correlation
- Optimization of boundary conditions for a mixing device
- Six Sigma analysis

Additional Optional Examples:

- What-if analysis
- Parameter correlation
- Design of experiments (DOE)
- Response surface-based optimization (bracket support)
- Direct optimization — Example No. 1
- Response surface-based optimization (cylindrical support)
- Direct optimization — Example No. 2
- Optimization based on a CFX model (airfoil)
- Sparse grid method
- Using a Python journal in optimization tasks

Specialized Course. ANSYS Fluent/Mechanical for Fluid–Structure Interaction (FSI) Simulation

Duration — 2 days

The course is focused on developing practical skills in modeling fluid–structure interaction (FSI), including the interaction between fluid flow (liquids and gases) and structures.It covers both one-way and two-way data coupling between CFD and structural analysis modules, as well as conjugate heat transfer simulations.

The course requires knowledge at the level of basic courses in ANSYS Fluent, DesignModeler, and ANSYS Meshing. Experience with ANSYS Mechanical or ANSYS Structural is recommended.

Course Outline:

- Introduction to FSI
- Overview of system coupling
- Workflow in Workbench for FSI simulation
- Settings for Mechanical, Fluent, and System Coupling modules
- Dynamic mesh model description
- Solution process for coupled problems and results analysis
- Convergence techniques for FSI simulations
- One-way FSI analysis

Standard Examples:

- One-way FSI analysis of a sensor with pressure field transfer
- Two-way FSI analysis of a hyperelastic plate
- Debugging FSI simulations
- Two-way FSI analysis of a fuel injector
- Thermal stress analysis of a T-junction pipe

Specialized Course.
Acoustic Analysis in ANSYS Mechanical

Duration — 2 days

The course covers both theoretical and practical aspects of acoustic simulation using Workbench Mechanical.It includes the creation of acoustic domains, interaction between acoustic media and structures, natural frequency extraction, harmonic and spectral analyses, as well as transient simulations.

The practical part features simulation problems involving mufflers, loudspeakers, and other engineering structures.

Course Outline:

- Introduction to Acoustics
- Modal Analysis
- Harmonic Analysis

Standard Examples:

- Modal analysis of air in a vehicle cabin
- Sloshing of liquid in a tank
- Loudspeaker and plate
- Sound scattering by a submarine
- Absorptive muffler
- Use of the full admittance matrix

Specialized Course.
Multiphysics Analysis: Electromagnetics–Structural Coupling for Solving Magnetic Field–Structure Interaction Problems

Duration — 2 days

The course is focused on developing practical skills in modeling the interaction between magnetic fields and deformable structures.

It covers both one-way and two-way iterative data exchange algorithms between ANSYS electromagnetic and structural analysis modules.

The course requires knowledge at the level of basic courses in ANSYS Maxwell 2D/3D, DesignModeler, and ANSYS Meshing. Experience with ANSYS Mechanical is recommended.

Magnetic field problems are solved using ANSYS Maxwell 2D/3D solvers.

Structural analysis is performed using ANSYS Static Structural and ANSYS Transient Structural.

Course Outline:

- Overview of coupled systems to help users select the most suitable ANSYS module for structural analysis
- Solving the electromagnetic problem in ANSYS Maxwell to determine sources of volumetric forces and moments
- Enabling mesh deformation in the electromagnetic model during iterative recalculation
- Workflow in ANSYS Workbench for solving multiphysics problems
- Mesh preparation in ANSYS Meshing
- Setup of ANSYS Static Structural and ANSYS Transient Structural modules
- Element-wise transfer of volumetric heat generation; solution process for coupled problems and results analysis
- Automatic iterative coupling of electromagnetic and structural simulations
- One-way coupling

Standard Examples:

- IGBT (Insulated Gate Bipolar Transistor)
- Current-carrying structural components
- User-defined problems

KAZAKH engineering

Specialized Course.
Multiphysics Analysis: Electromagnetics–Thermal Coupling for Cooling of Electronic and Electromechanical Devices

Duration — 3 days

The course is focused on developing practical skills in modeling heat transfer processes through the interaction of fluid flow (liquids and gases) with structures.

It covers both one-way and two-way iterative data exchange algorithms between ANSYS CFD and electromagnetic modules.

The course requires knowledge at the level of basic courses in ANSYS Maxwell 2D/3D, DesignModeler, and ANSYS Meshing. Experience with ANSYS Fluent or ANSYS IcePak is recommended.

Electromagnetic problems are solved using ANSYS Maxwell 2D/3D solvers.

Thermal analysis is performed using ANSYS Thermal.

For conjugate heat transfer simulations, ANSYS IcePak or ANSYS Fluent may be used.

Course Outline:

- Overview of coupled systems to help users select the most suitable ANSYS module for thermal analysis
- Solving the electromagnetic problem in ANSYS Maxwell to determine heat sources: ohmic losses, eddy current losses, and core (iron) losses
- Use of temperature-dependent properties to update the electromagnetic model during iterative recalculation
- Workflow in ANSYS Workbench for solving multiphysics problems
- Mesh preparation in ANSYS Meshing or ANSYS IcePak
- Setup of ANSYS Thermal, ANSYS Fluent, and ANSYS IcePak modules
- Element-wise transfer of volumetric heat generation; solution process for coupled problems and results analysis
- Automatic iterative coupling of electromagnetic and thermal simulations
- One-way coupling

Standard Examples:

- Cooling of an electric motor and generator (forced convection)
- Induction heating of a workpiece (natural convection)
- Cooling of a current-limiting reactor
- User-defined problems

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