KAZAKH engineering

ANSYS Course Catalog 2026

Geometry and Finite Element Meshing

kz-engineering.com

KAZAKH engineering

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

KAZAKH engineering

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

KAZAKH engineering

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

KAZAKH engineering

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

KAZAKH engineering

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

KAZAKH engineering

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

KAZAKH engineering

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

KAZAKH engineering

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

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