KAZAKH engineering

# ANSYS Course Catalog 2026

## Fluid Dynamics and Heat Transfer

kz-engineering.com

# 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)

KAZAKH engineering

# 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

# 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

KAZAKH engineering

# 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

# 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.

# Контакты

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