Welcome to the ANSYS North America training guide. In addition to the quality of the code, it is the training and support services we provide that have made ANSYS® the world leader it is today. We offer a wide range of training courses, for both beginners and more advanced users, covering the entire ANSYS product range. As well as providing frequently scheduled courses for all our software products, we also find many customers benefit from our advanced training courses or our ability to customize a class to focus on their particular applications. Our courses can either be delivered from our training rooms around North America, or delivered at a client’s premises, where we can supply additional software licenses or computers for the trainees. Please contact the training coordinator to discuss your particular needs and objectives.

For the latest information on course dates and course pricing, please visit our website www.ansys.com/support/training+center

For more information
Tel: 603-727-5405
Email: trng-admin@ansys.com

(c) 2013 ANSYS Inc. All rights reserved
All our training courses are delivered by engineers experienced both in our software and in the industries we serve, indeed many of whom hold PhDs in their specialty. We continually monitor and improve our courses to make sure we achieve and maintain the highest standards.

The facts speak for themselves. The figure below shows the collated feedback from our training courses. Trainer knowledge, helpfulness and presentation quality all combine to make the training sessions highly beneficial for all attendees.

We realise that everyone’s time is precious, and so it is of paramount importance that training gives our customers all the skills they need so they can go on to perform efficient, reliable and trustworthy analyses on their own.
CONTENTS

4 Fluids & Structural
  • ANSYS® DesignModeler™
  • ANSYS® SpaceClaim™

8 Fluid Dynamics
  • ANSYS® Meshing Platform
  • ANSYS® Fluent® Solver
  • ANSYS® CFX® Solver
  • ANSYS® ICEM CFD™
  • ANSYS® Fluent Meshing
  • ANSYS® Icepak®
  • ANSYS® DesignModeler for Electronics & ANSYS® CFDPost
  • Specialised CFD Training

14 Structural Mechanics
  • ANSYS® Mechanical™ Solver
  • ANSYS® DesignModeler and Meshing for Explicit Dynamics and Autodyne
  • ANSYS® Explicit STR / Autodyne® Part I
  • ANSYS® Autodyne® Part II
  • Specialised Mechanical Training

19 Offshore
  • ANSYS® Mechanical™ for Offshore Structures
  • ANSYS® Aqwa™

21 Electromagnetics
  • ANSYS® Maxwell®
  • ANSYS® HFSS™
  • ANSYS® HFSS® Advanced
  • PowerArtist™
  • RedHawk™
  • Totem™
  • Sentinel™
ANSYS™ DesignModeler® 1 day

**ANSYS® DesignModeler™ is our core geometry creation and modification tool.**

**Prerequisites**
The course assumes you have no prior experience of working with any CFD/FEA package, although a background in engineering/physics is assumed.

**Learning Objectives**
The ANSYS DesignModeler training course is for users who want to create and modify geometry in preparation for CFD or FEA analysis. Trainees who attend this course will learn:

- How to create and modify geometry in preparation for analysis
- How to navigate within the graphical user interface
- How to generate 2D sketches and convert them into 2D or 3D models
- How to modify 2D and 3D geometry
- How to import existing CAD geometry
- How to modify and clean up imported CAD
- How to model assemblies
- How to utilise parameters

Each course chapter is followed by “hands-on” workshops and exercises.
ANSYS® SpaceClaim® Direct Modeler 1 day

ANSYS® SpaceClaim® Direct Modeler is a feature-based geometry creation and modification tool.

Prerequisites
The course assumes you have no prior experience of working with any CFD/FEA package, although a background in engineering/physics is assumed.

Learning Objectives
The ANSYS SpaceClaim training course is for users who want to create and modify geometry in preparation for CFD or FEA analysis. The feature-based workflow of SpaceClaim provides an alternative workflow to that of the history-based ANSYS DesignModeler. On this course, trainees will learn how to create 3D geometry in SpaceClaim, how to work with assemblies, and how to simplify and repair externally-created geometry.

Each course chapter is followed by “hands-on” workshops and exercises.
ANSYS® Meshing 1 day

The ANSYS® Meshing is the front end interface for driving our meshing tools. You can select from and combine a variety of meshing methods to suit the nature of the geometry and the physics to be solved.

Prerequisites
Use of the ANSYS Meshing requires a geometric model produced either with ANSYS DesignModeler or third-party CAD software. Trainees should be familiar with the relevant CAD tool, for example, by attending the ANSYS DesignModeler course on the previous day so that creation/modification can be done prior to meshing.

Learning Objectives
By the end of this course you should be able to:

- Work with ANSYS Meshing, with full understanding of its GUI
- Understand the different meshing methods available for 2D and 3D geometries
- Create tetrahedral meshes
- Create hexahedral meshes
- Create inflation (boundary) layer meshes near to walls
- Apply advanced controls so as to refine and coarsen the mesh in different regions of the domain
- Examine the quality of the mesh.

Hands-on workshops are provided so trainees can apply the skills learnt.

Example of all-hex meshing in the automotive industry

Hex meshing of a cell phone
ANSYS® Fluent® Solver 2 days

ANSYS® Fluent® software provides a powerful, general-purpose CFD (Computational Fluid Dynamics) capability, incorporating a broad range of physical modelling.

Most new ANSYS Fluent users spend 4 days on introductory training:

Prerequisites
The two-day ANSYS Fluent course assumes no previous CFD knowledge, but that users have geometry and mesh creation skills (typically from attending our ANSYS DesignModeler and ANSYS Meshing courses on the preceding days).

Learning Objectives
This course will show how to set up a range of simulation types in the ANSYS Fluent solver, including how to:
- Define material properties
- Prescribe boundary and operating conditions
- Set solver controls
- Set up solution monitors
- Provide an initial solution
- Display and examine results
All topics are covered using a combination of lectures and hands-on sessions.

Recommended Follow-on Courses
See later page in this booklet on Advanced CFD Training.
**ANSYS® CFX® Solver** 2 days

**ANSYS® CFX® software provides powerful general-purpose CFD (Computational Fluid Dynamics) capability, incorporating a broad range of physical modelling.**

Most new ANSYS CFX users spend 4 days on introductory training:

- ANSYS DesignModeler (1 day)
- ANSYS Meshing (1 day)
- ANSYS CFX (2 days)

**Prerequisites**

Previous experience of CFD is not essential for the two day CFX Course. However, attendees should know how to import or create geometry and generate a suitable mesh, for example, by attending our ANSYS DesignModeler & ANSYS Meshing courses on the preceding days.

**Learning Objectives**

This course will show how to set up a range of simulation types in the ANSYS CFX solver, including how to:

- Select appropriate physical models
- Define material properties
- Prescribe boundary and operating conditions
- Set solver controls
- Set up solution monitors
- Use the ANSYS CFX Expression Language (CEL)
- Display and examine results

All topics are covered using a combination of lectures and hands-on sessions.

**Recommended Follow-on Courses**

See later page in this booklet on Advanced CFD Training.

Streamlines in a pump impeller
ANSYS® ICEM CFD™ 3 days

ANSYS® ICEM CFD™ is an advanced pre-processing tool, which incorporates a range of CAD readers and repair tools, along with advanced tetrahedral and hexahedral meshing capabilities.

Prerequisites
The two-day ANSYS ICEM CFD course assumes no previous geometry and mesh creation experience. However past exposure in the use of basic geometry and meshing tools would be beneficial.

Learning Objectives
The full ANSYS ICEM CFD training course is covered over two days, however trainees can choose to attend just one day depending on the type of meshing they require. Day 1 covers the import and repair of geometry, the creation of tetrahedral meshes with prism layers using ANSYS ICEM CFD Tetra/Prism and checking and smoothing the mesh. Day 2 the techniques for generating structure, hexahedral meshes with ANSYS ICEM CFD Hexa are taught.

All topics are covered using a combination of lectures and hands-on sessions:
- Geometry repair/creation
- Tetra/Prism meshing
- Hexa meshing
- Hexa advanced features
- Mesh editing

Pure hexahedral meshing
**ANSYS® Fluent® Meshing** 2 days

**ANSYS® Fluent® Meshing software provides powerful unstructured meshing capabilities where the starting point is a tessellated geometry input. A range of CAD readers are available, surface meshing and repair tools, along with hybrid and cutcell volumetric meshing capabilities.**

**Prerequisites**
The two-day ANSYS Fluent Meshing course assumes past exposure to the use of basic geometry and meshing tools.

**Learning Objectives**
ANSYS Fluent Meshing training takes place over two days (although some trainees with previous TGrid experience may choose to only attend day 2). On day 1, trainees learn how to import and repair surface meshes in ANSYS Fluent Meshing, along with the creation of hybrid and cutcell meshes. Day 2 is focused on CAD reader technology for direct CAD import and use of advanced wrapping techniques for surface mesh generation from assemblies of solid parts.

All topics are covered using a combination of lectures and hands-on sessions:
- Boundary mesh repair/improve tools
- Hybrid meshing
  - Prism/Tetra
  - Prism/Hexcore
- Cutcell Meshing
- CAD import
- Surface wrapping techniques
  - Wetted region extraction
  - Multi solid/fluid surface meshing
- Volume mesh repair/improve tools

Hexcore mesh inside an automobile cabin
ANSYS® Icepak® 3 days

ANSYS® Icepak® combines advanced solver technology with robust meshing options designed to provide fast and accurate thermal results for electronics cooling applications.

Prerequisites:
A technical education and background is recommended but an engineering degree is not required.

Learning Objectives:
This is the standard introductory course for those new to ANSYS Icepak. The primary goal of this course is to cover the basics of working in the ANSYS Icepak user environment. All topics are covered using a combination of lectures and hands-on sessions.

- Introduction to Icepak
- Model building with Icepak
- Icepak objects – primitives and compound objects
- Meshing
- CAD and ECAD import options within Icepak
- Physics: Fluid flow, heat transfer, steady state and transient analyses
- Defining a parametric study in Icepak: standard and Boolean parameters
- Solution set up
- Post-processing option
- Workbench integration

Graphics card heat sink, Fan rotating using MRF model
ANSYS® DesignModeler™ for Electronics & ANSYS® CFDPost 1 day

This is an advanced course designed for existing Icepak users and consists of two sessions. ANSYS® DesignModeler for Electronics is a specialised pre-processor used to simplify and transfer geometry created from CAD or within ANSYS DesignModeler to Icepak.

ANSYS® CFDPost provides more advanced post-processing functionality to that provided within ANSYS Icepak.

Prerequisites:
The course assumes no previous geometry creation experience, however past exposure to ANSYS Icepak would be beneficial.

Learning Objectives:
The course takes place over two sessions. In the first session, trainees learn how to import or create geometry in ANSYS DesignModeler and simplify the model to transfer to Icepak within ANSYS Workbench. In the second session, trainees will learn how to export data from ANSYS Icepak to ANSYS CFDPost, and how to postprocess models within ANSYS CFDPost.
Specialized CFD Training

We offer a range of advanced training solutions for CFD users. Below is a list of the courses that are commonly requested. As an alternative, you may find that a more tailored course (a ‘Focus Day’) better meets your needs.

<table>
<thead>
<tr>
<th>Course</th>
<th>ANSYS CFX</th>
<th>ANSYS Fluent</th>
</tr>
</thead>
<tbody>
<tr>
<td>Acoustics</td>
<td>✓</td>
<td></td>
</tr>
<tr>
<td>Automotive HVAC</td>
<td>✓</td>
<td></td>
</tr>
<tr>
<td>Automotive Underhood</td>
<td>✓</td>
<td></td>
</tr>
<tr>
<td>Combustion / Reacting Flows</td>
<td>✓</td>
<td>✓</td>
</tr>
<tr>
<td>Customisation of CFX</td>
<td>✓</td>
<td></td>
</tr>
<tr>
<td>Dynamic Mesh Modelling</td>
<td></td>
<td>✓</td>
</tr>
<tr>
<td>Heat Transfer</td>
<td></td>
<td>✓</td>
</tr>
<tr>
<td>Multiphase Modelling</td>
<td>✓</td>
<td>✓</td>
</tr>
<tr>
<td>Fluid-Structure Interaction</td>
<td>✓</td>
<td>✓</td>
</tr>
<tr>
<td>Spray and Particle Modelling</td>
<td></td>
<td>✓</td>
</tr>
<tr>
<td>Turbomachinery</td>
<td>✓</td>
<td>✓</td>
</tr>
<tr>
<td>Turbulence Modelling</td>
<td>✓</td>
<td>✓</td>
</tr>
<tr>
<td>User Defined Functions</td>
<td></td>
<td>✓</td>
</tr>
</tbody>
</table>

Focus Days:
Focus days offer flexible, one-to-one coaching with an experienced engineer and can be based on an example of the customer’s specific application. For some users, they are an invaluable way of improving productivity through optimisation of meshing or model-building techniques. Other users may have problems with complex underlying physics which will benefit from expert advice on how to implement and solve the appropriate models.

Vortices behind Aircraft Landing Gear
ANSYS® Mechanical™ Solver

ANSYS® Mechanical™ is our core FEA (Finite Element Analysis) software for the simulation of linear and non-linear static and dynamic systems.

Prerequisites
A technical education and background is assumed but no previous experience of FEA is needed.

Learning Objectives
This course will give a good overview of the interface and several analysis types:
- General Pre-processing, Contact and Meshing
- Static Structural
- Modal
- Steady State Thermal
- Linear Buckling
- Post Processing
- How to use CAD and run parameter studies
- Named Selection
- Remote Boundary Conditions
- Joints, Beams and Springs
- Virtual Topology
- Rigid Bodies
- Constraint Equations
- Multistep Analysis
- Contact and Mesh Connections

Recommended Follow-on Courses
See later page in this booklet on Advanced Mechanical Training.
ANSYS® DesignModeler™ and Meshing for Explicit STR and Autodyne® 2 days

Prerequisites
The course assumes you have no prior experience of working with any CFD/FEA or mesh generation package, although a background in engineering/physics is assumed.

Learning Objectives
This training course is for users that want to create and modify geometry and create good quality numerical meshes suitable for explicit dynamics simulations. The course will cover the following subjects:

ANSYS DesignModeler
- How to create and modify 2D & 3D geometry
- How to import and clean up existing CAD geometry
- How to parameterise geometry
- How to prepare geometry for meshing

Meshing for ANSYS Explicit Dynamics & ANSYS Autodyne
- Overview of mesh methods for 2D and 3D problems
- Mesh controls for Explicit Analysis / ANSYS Autodyne
- Repairing / defeaturing CAD geometry to aid with meshing
- Block structured mesh methods to create good quality hexahedral swept meshes for a range of different geometries

Example of a swept hexahedral mesh generated using a combination of ANSYS DesignModeler and ANSYS Meshing
ANSYS® Explicit STR / Autodyne® Part I 2 days

ANSYS® Explicit STR / Autodyne® Part I extends the range of problems that can be simulated by ANSYS Mechanical to simulate short-duration severe loading and complex contact.

Prerequisites
A basic knowledge of dynamics and strength of materials (material modelling) is highly recommended. Knowledge of the physics of transient dynamics events is also recommended. A basic knowledge of ANSYS Mechanical in ANSYS Workbench is advantageous but not required. This course is generally considered to be a prerequisite to the ANSYS Autodyne 2 day training course.

Learning Objectives
This course is designed for new users who want to become proficient with ANSYS Explicit STR in ANSYS Workbench. You will focus on learning core-modeling skills in this comprehensive, hands-on course. After completing the course you will be well prepared to work effectively on a wide range of transient dynamics applications from low velocity drop tests through to high velocity ballistic impacts.

Topics covered include:
- Introduction to ANSYS Explicit STR and ANSYS Workbench
- Explicit dynamics basics
- Results processing
- Explicit meshing
- Body interactions
- Material modelling
- Optimisation studies

Highly nonlinear crushing of a drinks can
ANSYS® Autodyne® Part II 2 days

ANSYS® Autodyne® Part II software is a versatile explicit analysis tool for modeling the nonlinear dynamics of solids, fluids, gases and their interactions.

Prerequisites
A basic knowledge of dynamics and strength of materials (material modeling) is highly recommended. A knowledge of the physics of transient dynamic events is also recommended. It is advised that participants of this 2-day course should have already attended the ANSYS Explicit STR / Autodyne Part I training which is generally run in the 2 days prior.

Learning Objectives
This course is designed for users who are familiar with ANSYS Explicit STR in ANSYS Workbench who want to become proficient with the advanced options available in ANSYS Autodyne. You will focus on learning core-modeling skills in this comprehensive hands-on course. After completing the course you will be well prepared to work effectively using ANSYS Explicit STR and ANSYS Autodyne on a wide range of transient dynamics applications from hypervelocity impact through to blast modeling.

Topics covered include:
- Autodyne user interface
- Problem setup
- Autodyne in workbench
- Euler solvers
- ALE solver
- Mesh-free solver
- Advanced material models
- Euler-Lagrange interactions
- Parallel processing

Bird stike on aircraft wing structure

Blast propagation over a ship generated by an explosion on the waterline
Specialized Mechanical Training

We offer a range of advanced training solutions for ANSYS Mechanical users. Below is a list of the set courses that are commonly requested. As an alternative, you may find that a more tailored course (a ‘Focus Day’) better meets your needs.

Advanced Training Topics (ANSYS Mechanical)
- Advanced Structural Nonlinearities and Contact
- Heat Transfer
- Linear and Non-Linear Dynamics
- Using APDL Command Objects
- Rigid Body Dynamics
- Other advanced topics as needed

Advanced Training Topics (ANSYS Explicit tools)
- Material Modeling: ductile & explosive
- Material Modeling: brittle & composite
- User subroutines
- Explicit Dynamics with ANSYS LS DYNA

Focus Days
Focus days offer flexible, one-to-one coaching with an experienced engineer and can be based on an example of the customer’s specific application. For some users, they are an invaluable way of improving productivity through optimisation of meshing or model-building techniques. Other users may have problems with complex underlying physics which will benefit from expert advice on how to implement and solve the appropriate models.
ANSYS® Mechanical™ for Offshore Structures 2 days

ANSYS® Mechanical for Offshore Structures is designed to introduce the process required to carry out the analysis of Offshore Jacket type structures.

Prerequisites
This course is an introductory course, therefore no previous ANSYS Mechanical experience is necessary.

Learning Objectives
This course centers on practical workshops combined with tutorial sessions in order to give advice and useful experience for creating analyses in an efficient and effective manner, and, understanding the process for offshore design using ANSYS Mechanical.

Topics covered include:
- Mechanical Basics
- General Preprocessing
- Geometry Modeling for Offshore Structures
- Ocean Loading on PIPE elements
- Pile Analysis
- Design Checks
- Fatigue Analysis

Image Courtesy of REpower Systems AG
ANSYS® AQWA™

ANSYS® AQWA™ software is used for the hydrodynamic assessment off all types of offshore and marine structures.

Prerequisites
No previous ANSYS AQWA exposure is required but some knowledge of naval architecture or offshore engineering is desirable. The course will also be of benefit to existing ANSYS AQWA users both to introduce the many new capabilities/features and as a refresher course.

Learning objectives
The course will show you how to create your own ANSYS AQWA analyses efficiently and effectively. All topics are covered using a combination of lectures and hands on sessions. Topics covered include:

- An introduction to ANSYS Workbench
- Geometry creation using ANSYS DesignModeler
- Mesh generation
- Hydrodynamic diffraction and radiation analysis
- Definition of mooring systems, articulations and fenders
- Frequency domain analysis
- Determination of structural equilibrium position under the influence of environmental conditions and constraint such as mooring lines and articulations.
- Static and dynamic (oscillatory) stability
- Time domain analysis in regular and irregular waves for moored structures
- Multi-body hydrodynamic interaction
- Parametrics
- Load mapping to ANSYS Mechanical
Maxwell® 2 days

Maxwell® software is for designing and analysing 3-D and 2-D electromagnetic and electromechanical devices such as motors, actuators, transformers, sensors and coils.

Prerequisites
This training is for research, design, instrumentation and measurement engineers. A general knowledge of electromagnetic phenomena is recommended. There are no required prerequisites for experience in using simulation software.

Course Description
The course will teach students how to effectively use ANSYS Maxwell to setup, solve and post-process the results from electromagnetic and electromechanical models. The two-day training course will cover the following topics:

- Maxwell overview
- Introduction to the Finite Element Method
- Electrostatics
- DC Conduction
- Magnetostatics
- Parametric Modelling
- Transient Simulations
- Post-Processing
- Eddy Currents
- Optimetrics

Each topic is followed by hands-on workshops and exercises.

Flux density and flux line plot on a toothed wheel for an ABS sensor application
HFSS™ 1 day

Prerequisites
This introductory course is designed for users who are new to HFSS or the finite element method used in the tool. It is assumed that attendees have a basic knowledge of electromagnetics to include the following topics: Maxwell’s equations, transmission line theory and interpretation of propagation constant, waveguide theory and waveguide modes, characteristic impedance, S-Parameters.

Users should also be familiar with 2D/3D drawing tools and concepts although basic training is provided.

Learning Objectives
After completing this course, you should be able to:

- Draw and define geometry
- Assign materials
- Define ports and boundaries appropriately
- Set solver controls
- Solve basic models
- Display and examine results.

Course Agenda
All topics are covered using a combination of lectures and hands-on sessions.

Flux density and flux line plot on a toothed wheel for an ABS sensor application
**PowerArtist™ 1 day**

**PowerArtist** is ANSYS® Apache’s complete RTL design-for-power platform with fully-integrated advanced analysis and automatic reduction technologies; delivering 10% to 60% or more power savings.

**Prerequisites**
This hands-on course is designed for new or current users who may want to refresh their skills.

**Learning Objectives**
The course will cover the following topics:
- PowerArtist RTL design for power methodology
- RTL and gate-level power analysis
- Interactive power debug
- Automatic RTL power reduction
- Power regressions and data mining
- Display and examine results.

**Course Agenda**
All topics are covered using a combination of lectures and hands-on sessions.

**RedHawk™ 1 day**

**RedHawk** allows designers to explore and identify physical design weaknesses (RHE), automatically repair supply noise source (FAO), analyze the impact of dynamic voltage drop on timing and jitter (PSI), verify power and signal EM (SEM), validate ESD protection robustness (PathFinder™), provide a power delivery network model profile for system-level analysis (CPM™), and allow modeling, simulation and debug of 3D-IC designs (MDO).

**Prerequisites**
This course is designed for new or current users who may want to refresh their skills.

**Learning Objectives**
The course will cover the following topics:
- RedHawk flow overview, including how to debug designs using RedHawk Explorer
- Static IR/EM analysis including What-If analysis
- SoC-level SignalEM analysis
- Dynamic Vectorless/VCD Analysis
- Early analysis + Automatic FA
**Totem™ 1 day**

**Totem** is ANSYS® Apache’s full-chip, layout-based power and noise platform for analog and mixed-signal designs

**Prerequisites**
This course is designed for new or current users who may want to refresh their skills

**Learning Objectives**
The course will cover the following topics:
- Totem platform overview
- Totem-MMX flow overview
- Static IR/EM analysis including ‘What-If’ analysis
- Dynamic analysis
- Signal EM analysis with focus on dynamic signal EM flow
- Advanced Topics
  - Hierarchical analysis for large designs
  - Substrate analysis
  - Electro Static Discharge (ESD) verification

**Sentinel™ 1 day**

**ANSYS® Apache’s Sentinel** is a complete Chip-Package-System (CPS) co-design / co-analysis solution addressing system-level power integrity, I/O-SSO, thermal, and EMI challenges.

**Prerequisites**
This hands-on course is designed for new or current Sentinel-PSI users who may want to refresh their skills

**Learning Objectives**
The course will cover the following topics:
- Sentinel-PSI flow overview
- Layout import and editing
- Setup and analysis
  - DC IR-drop analysis
  - AC S parameter extraction
  - EMI analysis
  - Transient analysis
  - SSO Channel Builder and Sentinel-SSO integration
For the latest information on course dates and course pricing, please visit our website:
www.ansys.com/support/training+center

For more information
Tel: 603-727-5405
Email: trng-admin@ansys.com

(c) 2013 ANSYS Inc. All rights reserved

ANSYS, ANSYS Workbench, Autodyr, CFX, Fluent and any and all ANSYS Inc brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS Inc. or its subsidiaries in the United States or other countries. ICEM CFD is a trademark used by ANSYS Inc. under license. All other brand, product, service and feature names or trademarks are the property of their respective owners.