Training Catalogue

Introduction

Welcome to the Fluid Codes training catalogue

In addition to the quality of the Ansys products, the training and support provided by Fluid Codes makes Ansys the leader in engineering simulation across the EMEA region. Fluid Codes offers a wide range of training courses for the entire Ansys product range, from beginner to expert level users.

How can you reduce the learning curve by using our tools?

Fluid Codes' Training Services are developed for various disciplines of engineering, covering basic topics to the most advanced, to achieve simulation goals. Through our standard on site training courses, our solutions enable the participant to be successful in engineering using Ansys.

How can I request a training?

Visit www.fluidcodes.com and go to the "Training" section.

Table of Contents

Training Information

What type of training do we offer?

- **Standard Training:** focused on either introductory Ansys knowledge or a physic's specialization.
- **Advanced Training:** focused on advanced topics and their applications.
- **Customized Training:** focused and adapted to a specific application and simulation need.

Where can our training sessions take place?

- **In our offices**. In this case, we provide the complete infrastructure for conducting the training.
- **At Customer Site.** A Fluid Codes engineer will be sent to your company for the duration of the training period. We provide the temporary license keys and, optionally, computers.

What is the cost of the training?

- Our standard training costs **1800 USD/day*** and customized training costs approximately **2500 USD/day**, with a maximum of **8 computers**.
- Each computer can be shared between 2 participants.
- For Customized Training prices, you can contact our team at sales@fluidcodes.com

* In case of on-site training at the customer's location, this amount will be increased to cover the trainer's travel expenses.
Special discounts apply for academics.

Testimonials

Subject: Letter of Appreciation - Fluid Codes Ltd.

SABIC is a world leader in the petrochemical industry and is the largest listed company in the Middle East. SABIC manufacturing, sales, technology and innovation facilities are located throughout the globe.

SABIC recognizes the benefits of the state-of-the-art modeling tools such as ANSYS to enable breakthrough innovation and improve existing manufacturing assets safety, reliability and throughput. We actively use ANSYS software to optimize our chemical processes and to develop new chemical technologies and products.

We are pleased to inform you that the recent training sessions and ongoing technical support provided by Fluid Codes ANSYS channel partner for the Middle East have been excellent. Support from Fluid Codes expert team helps to enhance our in-house knowledge, make best use of ANSYS modeling tools, and add value to our business.

Sincerely,

Ramsey Bunama, Ph.D., P.Eng

Senior Manager, Advanced Technology Platform

RESEARCH & DEVELOPMENT CENTER
Oil & Gas Network Integrity R&D Division
P.O. Box 62, Bldg. 2297, Rm. GC-121, Dhahran
Phone: 876-1359; Fax: 876-7808
April 26, 2018

LETTER OF APPRECIATION:
FLUID CODES LTD.

To whom it may concern:

Saudi Aramco is the state-owned oil company of the Kingdom of Saudi Arabia and a fully integrated, global petroleum and chemicals enterprise. Over the past 80 years, Saudi Aramco has become the world largest integrated Oil and Gas Company. Saudi Aramco's scale of production, operational reliability, and technical advances, makes us the world's largest producer of crude oil and condensate.

For Saudi Aramco, Ansys is today a key tool that helps address numerous challenges e.g. rehabilitation of ageing assets, process optimization or maintenance activity. On a regular basis, Fluid Codes delivers services to Saudi Aramco and in particular, technical support, engineering services and customized trainings. Last February, a customized training was given to a team of modeling specialists at the Research Center of Dhahran, Saudi Arabia.

We are pleased to underline the high quality of this training. The content was extremely relevant as well as the dedication and knowledge of the trainer. The training was also ideally performed so users could immediately involve their new skills to the industrial application of interest.

Thibault Villette
Saudi Aramco
Team Leader, Modeling Team
Oil & Gas Network Integrity R&D Division
Research & Development Center
Email: thibault.villette@aramco.com
Tel: +966 13 872-0461

NPCC

We are happy to inform you that with the training `ANSYS Presure Vessel Modelling` held at 14-15 September, 2014 by and technical support given by Fluid Codes, NPCC has been able to apply ANSYS Mechanical to all kinds of pressure vessel design challenges in a quick and easy manner and our engineers have been able to quickly progress with the project requirements.

Bashar Yalchiner
Senior Engineer

EXTERRAN.

We are happy to inform you that with the training and support from Fluid Codes, Belleli Energy Srl has been able to apply ANSYS in a quick manner and our engineers have been able to quickly progress with the project requirements.

MASSIMILIANO CENTI
Head of Engineering

dar al-handasah
shair and partners

Generally, the training course was very useful in familiarizing DAR staff with CFD technology and the software capabilities. The course contents were relevant to our needs and covered most of the features of the software. In addition, the quality of the handouts was very good.

Maroun El Khoury
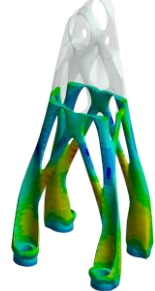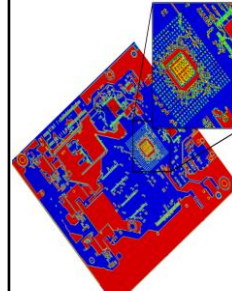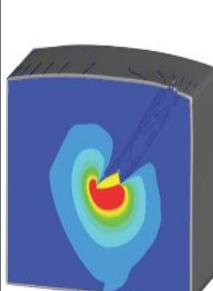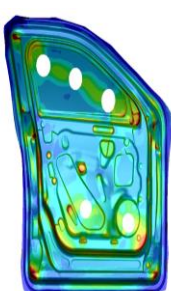Director of Mechanical Department
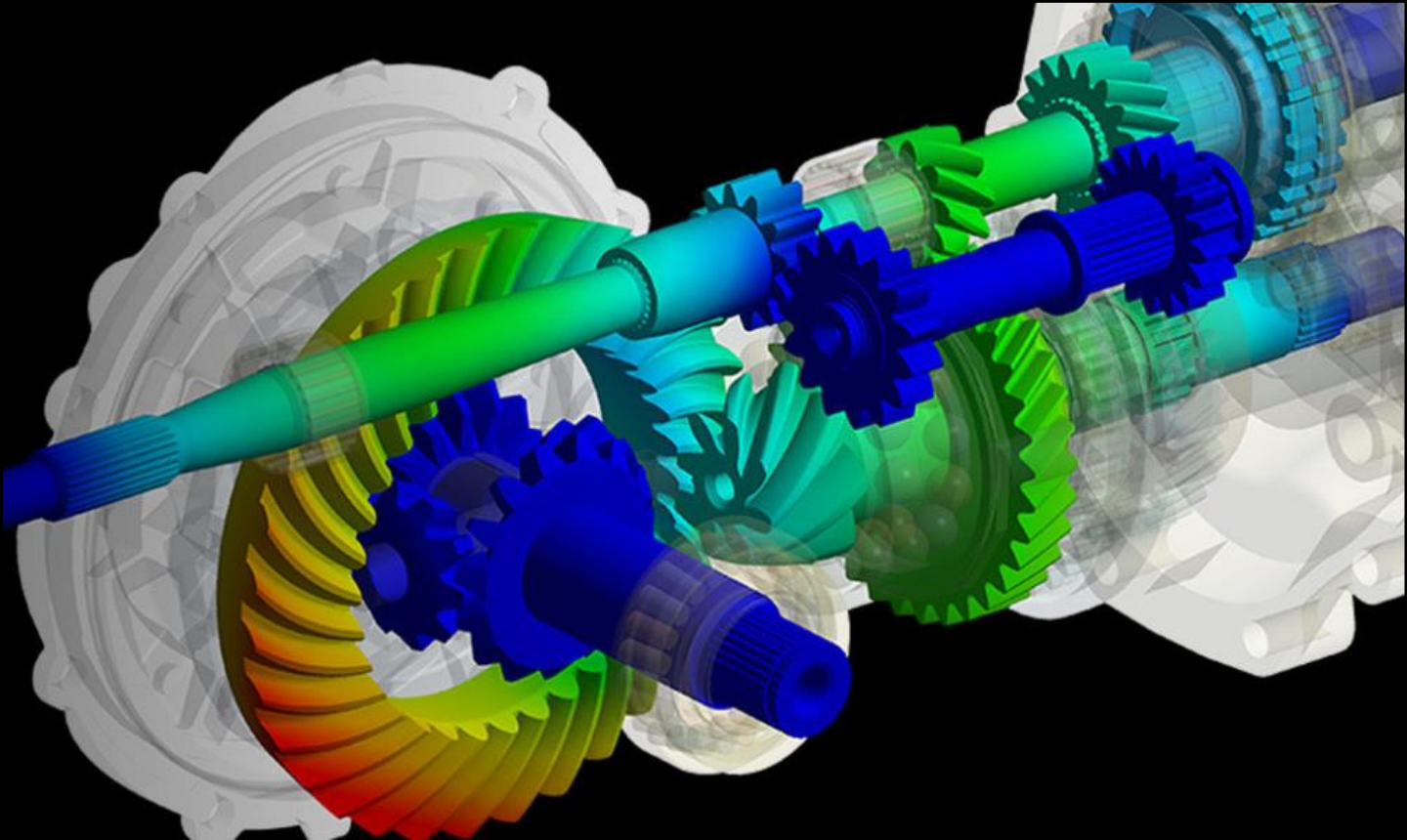
Structures

Mechanical	LS-DYNA	Forming	Auto-dyn	Sherlock	Additives
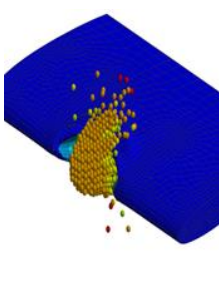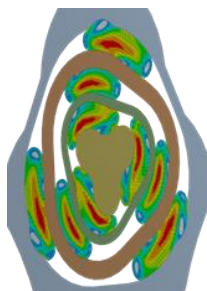
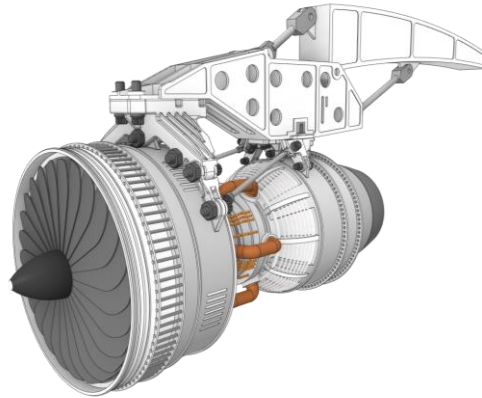# Geometry Preparation with Ansys

>> Duration

1 day

>> Participants

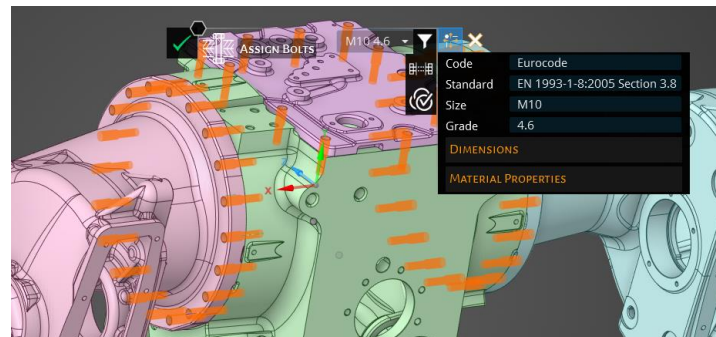Engineers and Designers

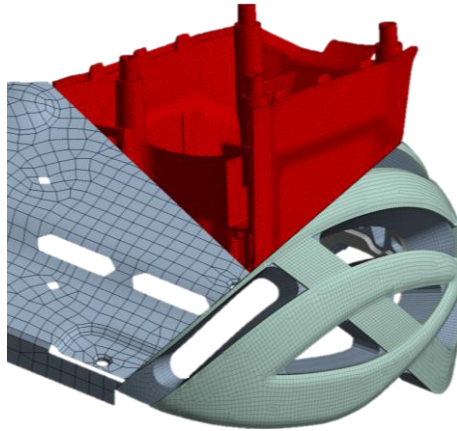## >> Prerequisites

Engineering knowledge

Overview

The Geometry Preparation course explores the basis of pre-processing using Ansys CAD-agnostic tool SpaceClaim. This ensures participants possess the knowledge to utilize Ansys core simulation tools such as Mechanical, Fluent, HFSS, Maxwell, etc. This training allows the participants to get acquainted with relevant features of SpaceClaim and its ability to prepare the domain for meshing in FEA or CFD simulations.

Topics

- Navigate within the GUI
- Generate 2D sketches and convert them into 2D or 3D models
- Selection basics
- Modify 2D and 3D geometry
- Import existing CAD Geometry
- Modify and clean up imported CAD
- Model assemblies
- Utilize parameters

Ansys Meshing for FEA

>> Duration

1 day

>> Participants

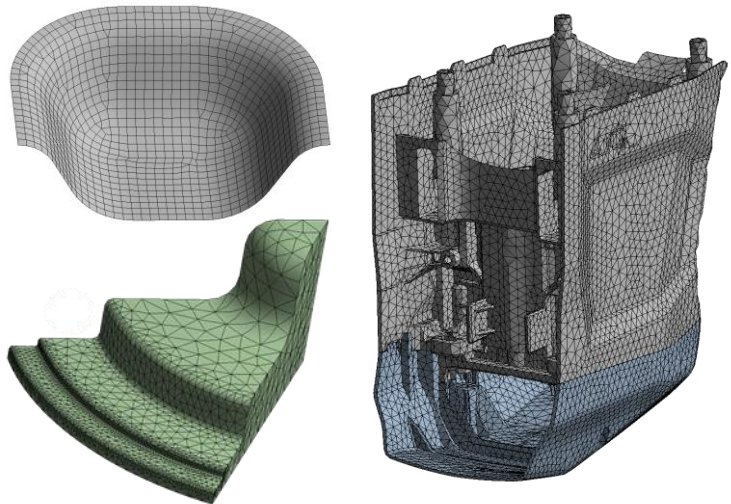Engineers and Designers

>> Prerequisites
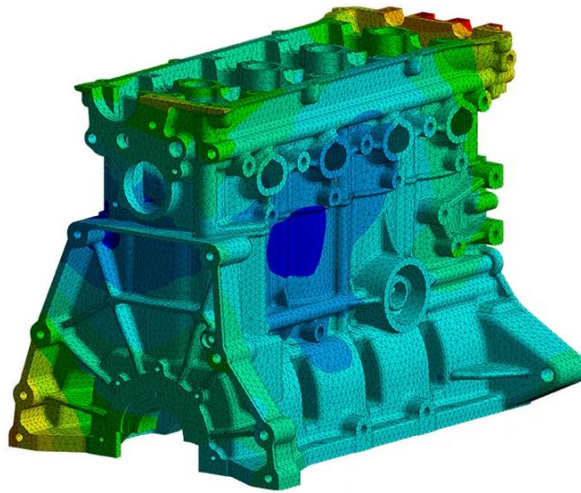
Engineering knowledge fundamentals – FEA

Overview

The Ansys Meshing for FEA course covers the principles of mesh generation along with its workflow. Meshing guidelines and best practices are at the core of this course. Ansys Meshing for FEA ensures participants have the pre-requisites to perform simulation in Ansys core FEA simulation tools such as Mechanical, LS Dyna, etc.

Topics

- Mechanical meshing core skills
- Meshing methods
- Global mesh controls
- Local mesh controls
- Mesh quality and advanced topics

Introduction to Ansys Mechanical

>> Duration

2 days

>> Participants

Engineers and Designers

>> Prerequisites

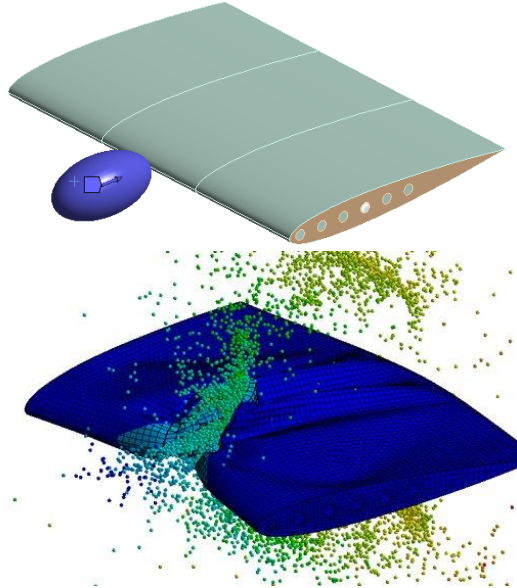Engineering knowledge fundamentals – FEA

Overview

This course offers participants the opportunity to develop their simulation skills in Finite Element Analysis (FEA) using Ansys Mechanical. Using Ansys Mechanical, participants will be able to analyze the behavior of components or assemblies under operating conditions and loads. The course provides a comprehensive introduction to the principles and practical applications of FEA, enabling participants to effectively utilize this technique for structural analysis. Additionally, participants will gain valuable experience in using Ansys Mechanical, a widely used software tool for FEA simulations.

Topics

- General pre-processing, contact and meshing
- Static structural
- Modal analysis
- Steady state thermal
- Post processing
- Rigid bodies
- Constraint equations
- Multistep analysis

Ansys Explicit Dynamics

Advanced
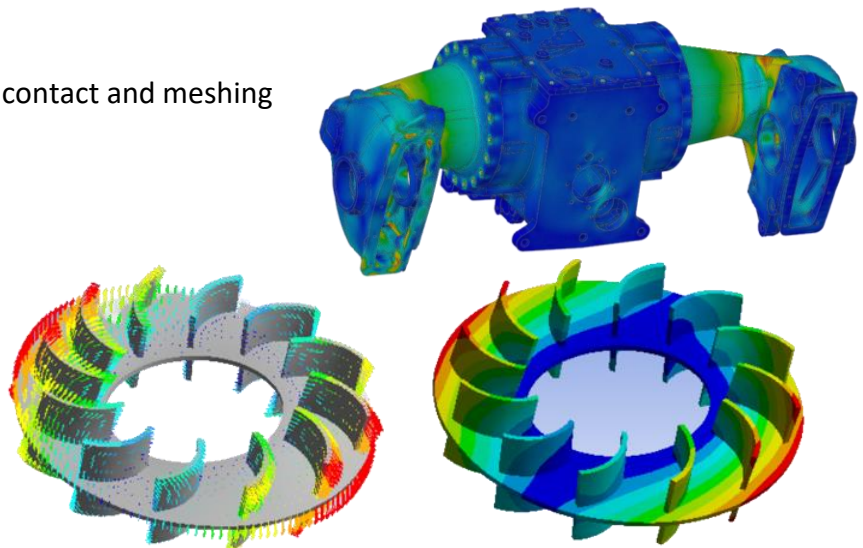
>> Duration

2 days

>> Participants

Mechanical Engineers,
Impact Dynamics Engineers,
Drop Test Engineers

>> Prerequisites

Engineering knowledge
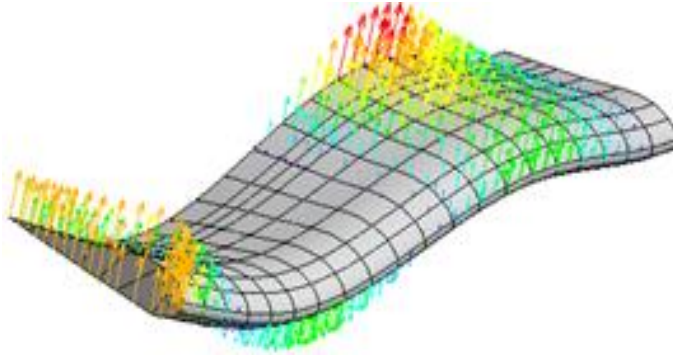fundamentals – FEA & Ansys
Mechanical

Overview

This course is targeted for engineers involved in design and analysis who want to simulate high-velocity dynamic problems such as blasts, explosions, and impacts. It provides a thorough understanding of the explicit solution method and its distinctions from other methods for dynamic analysis in Ansys. Participants will learn to select the appropriate method, apply explicit dynamic setup, create accurate meshing, and use proper solution settings. By the end of the course, participants will have the skills and knowledge needed to perform effective simulations for high-velocity dynamic problems using Ansys.

Topics

- Explicit Dynamics setup and workflow
- Introduction to Explicit
- Meshing in explicit dynamics
- Material models
- Connections
- Analysis settings configuration
- Euler and particle reference frames
- Post-processing Explicit results
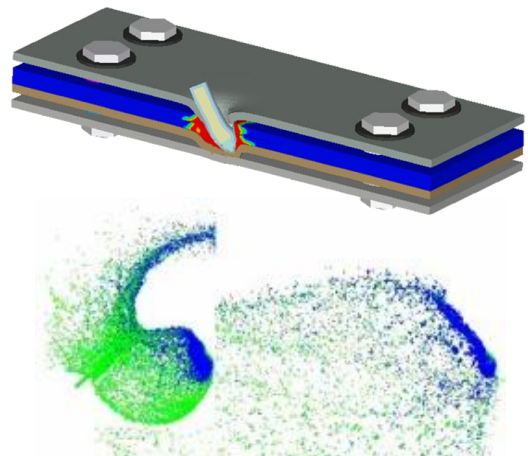
Ansys Linear & Non-Linear Dynamics

Advanced

Overview

This training course offers an overview of the various types of dynamic analyses available in Ansys Mechanical. It equips participants with the skills to select the most relevant analysis type for their needs and interpret the results accurately to take informed design decisions. Throughout this course, participants will gain a comprehensive understanding of the dynamic analysis capabilities of Ansys Mechanical, enabling them to apply this knowledge effectively to real-world scenarios.

Topics

- Introduction to dynamic analysis
- Modal analysis
- Linear perturbation
- Response spectrum analysis
- Harmonic analysis
- Random vibration analysis
- Transient analysis
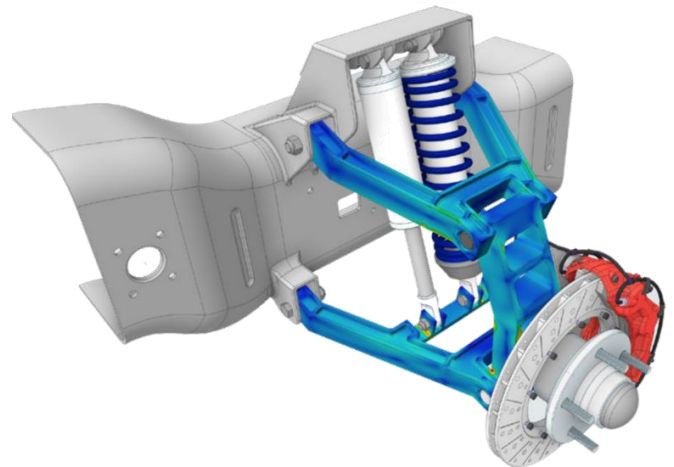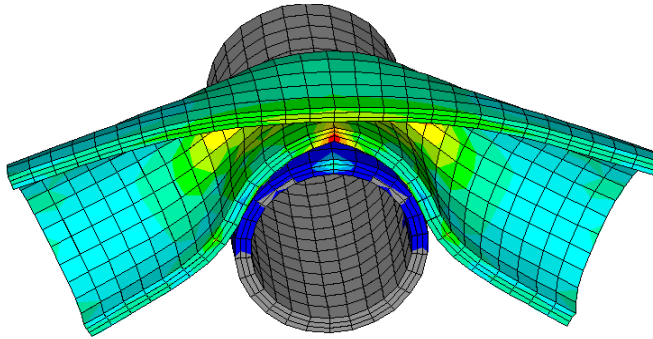
Ansys Structural Non-Linearities

Advanced

Overview

Throughout this training course, participants will acquire a thorough understanding of the nonlinear solution algorithm and procedures necessary for nonlinear simulations. They will learn to apply these techniques to tackle complex scenarios such as large deflection, surface-to-surface contact, elastic-plastic material models, and nonlinear buckling. The course will also provide valuable insights into identifying common causes of convergence difficulties in nonlinear solutions. The course's goal is to equip participants with the skills and knowledge required to perform effective nonlinear simulations and make informed design decisions in real-world scenarios using Ansys.

Topics

- Overview on structural non-linearities
- General procedure for non-linear simulations
- Large deflection
- Introduction to contacts
- Independent plasticity
- Buckling and linear perturbation
- Non-linear diagnostic
- Mesh nonlinear adaptivity
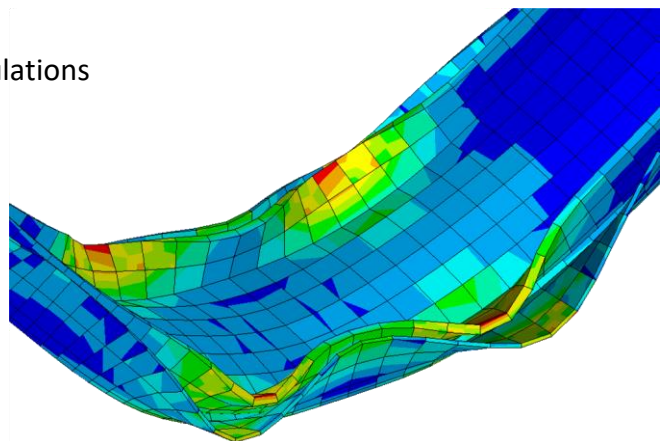
Ansys Non-Linear Contacts

Advanced

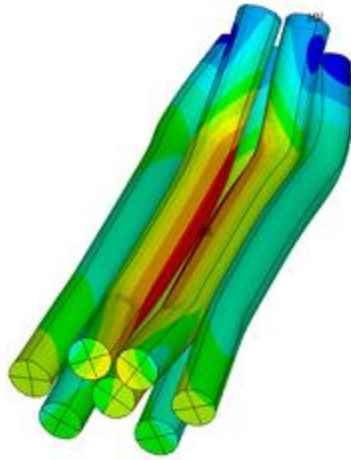## >> Duration

2 days

>> Participants

Mechanical Engineers, Design Engineers
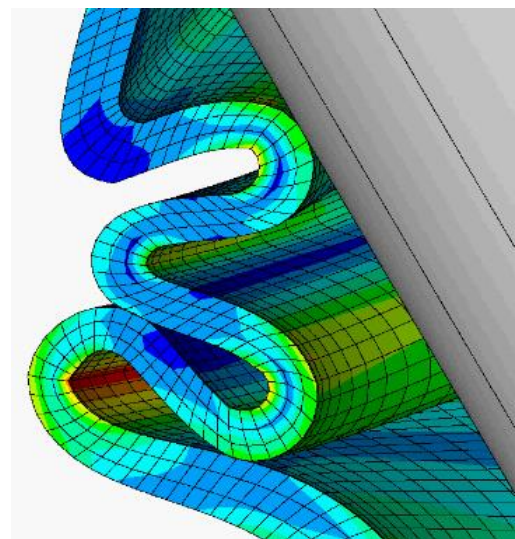
>> Prerequisites

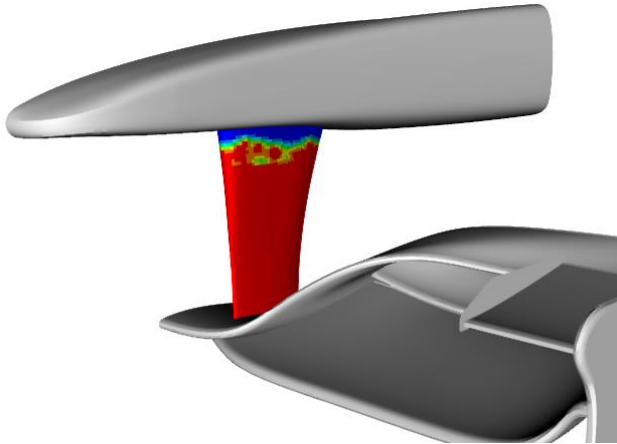Engineering knowledge fundamentals & experience with Ansys Mechanical

Overview

This training course covers advanced connections between structural systems, including non-linear elements like seals, gaskets, interference fittings, and bolt pretension. Participants will learn to identify frequent causes of convergence issues in non-linear solutions. The course equips participants with the skills to apply advanced connection techniques and tackle convergence issues in real-world scenarios.

Topics

- Contact formulations
- Detection methods
- Trim contact
- Penetration tolerance
- Contact stiffness
- Pinball region
- Symmetric and asymmetric contact
- Body types in contact
- Post-processing contact results

Ansys Non-Linear Materials

Overview

This advanced course aims to provide participants with knowledge and comprehension of material models used to represent non-linear material behavior in metals and elastomers. By the end of the course, participants will have a thorough understanding of these models and how to apply them to engineering scenarios.

Topics

- Advanced metal plasticity
- Creep
- Viscoplasticity
- Hyperelasticity
- Viscoelasticity
- Advanced models
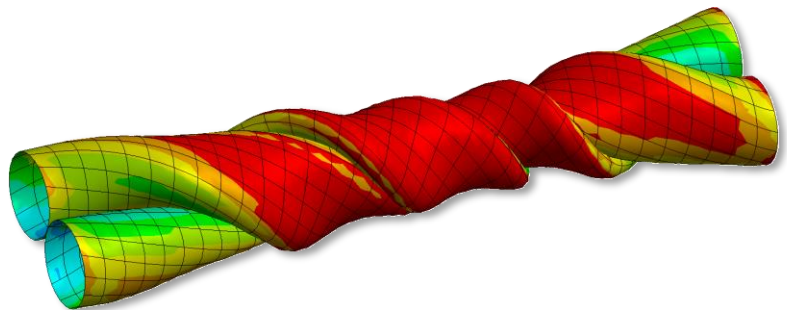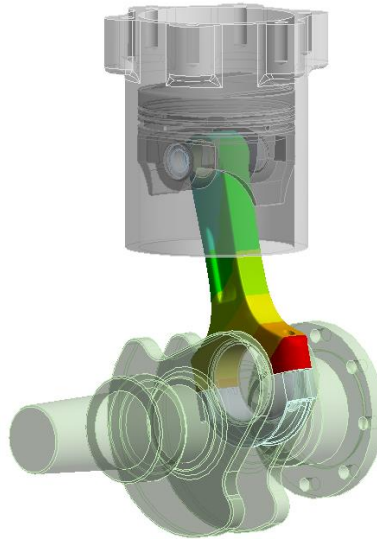
Ansys Rigid Body Dynamics

>> Duration

2 days

>> Participants

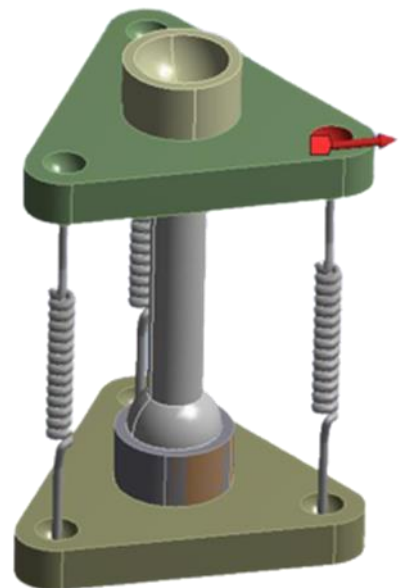Mechanical Engineers, Design Engineers

>> Prerequisites

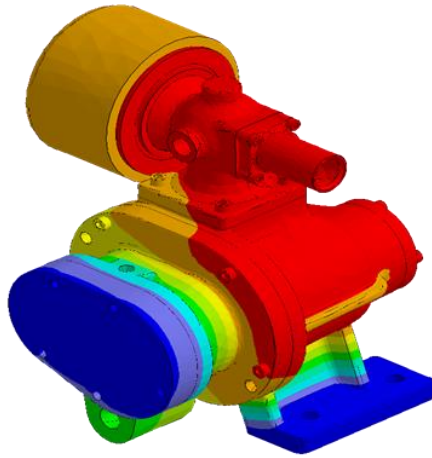Engineering knowledge fundamentals – FEA & Ansys Mechanical

Overview

This training course showcases the power of the rigid dynamics explicit solver in efficiently and robustly evaluating mechanical systems containing complex assemblies of interconnected rigid parts undergoing large overall motion. Participants will gain an understanding of how to utilize this solver to effectively analyze such systems and make informed design decisions.

Topics

- Introduction to rigid body motion
- Analysis configuration steps
- Connections
- Joints definition
- Rigid/flexible multibody dynamics
- Transient structural
- Link with control and system simulation

Ansys Thermal Analysis

Advanced

>> **Duration**

2 days

>> **Participants**

Mechanical Engineers, Design Engineers
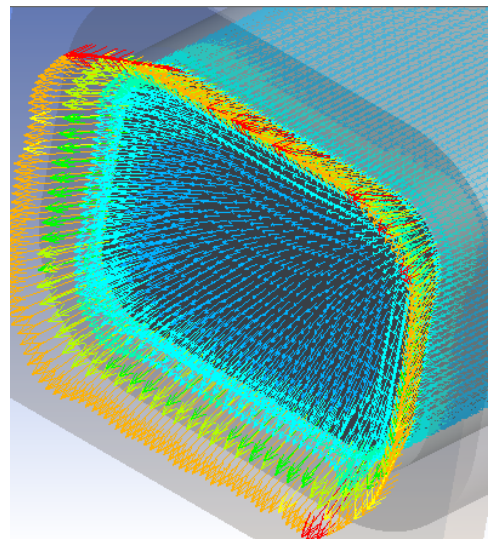
>> **Prerequisites**

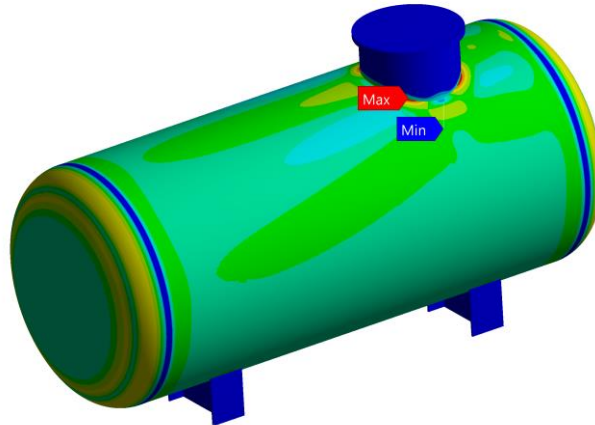Engineering knowledge fundamentals & experience with Ansys Mechanical

Overview

This course aims to equip students with the knowledge and skills necessary to evaluate a structure's thermal response to different heat loads. It covers methodologies for conducting complete solution processes for three heat transfer phenomena under steady-state and time-varying conditions, as well as thermal stress analysis. By the end of the course, participants will have a thorough understanding of how to assess thermal responses in structures and perform thermal stress analysis.

Topics

- Heat transfer fundamentals
- Preprocessing
- Boundary conditions
- Steady state heat transfer
- Non-linear thermal analysis
- Transient thermal analysis
- Advanced heat transfer
- Thermal-structural interaction

Ansys Fatigue Analysis Tool

>> Duration

2 days

>> Participants

Mechanical Engineers, Design Engineers including Vibration, automotive, industrial equipment engineers
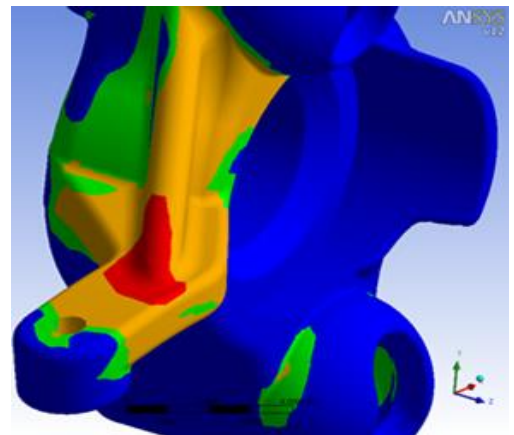
>> Prerequisites

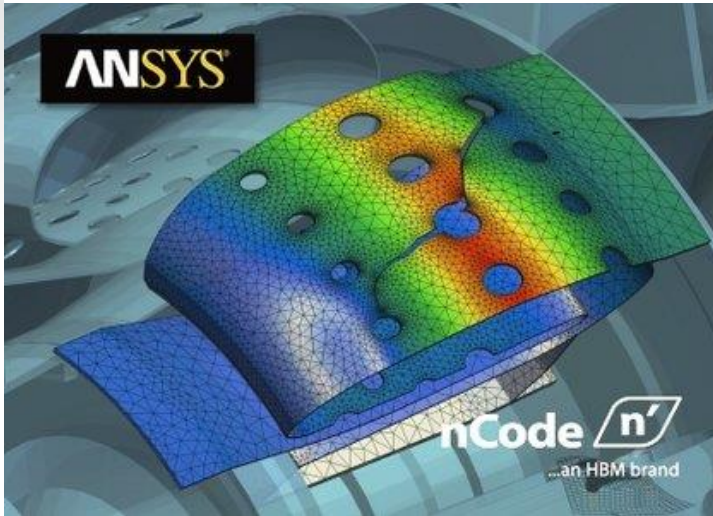Engineering knowledge fundamentals - Ansys Mechanical fatigue analysis

Overview

This advanced training course teaches students how to select the best type of analysis for a given problem by understanding stress-life and strain-life fatigue analysis methodologies. Participants will also learn about different types of fatigue loading on structures, including proportional, non-proportional, constant amplitude, and variable amplitude loadings. By the end of the course, participants will be equipped with the knowledge and skills necessary to effectively analyze fatigue in structures and make informed design decisions.

Topics

- Introduction to fatigue analysis
- High cycle fatigue
- SN curves
- Low cycle fatigue
- Fatigue according to ASME construction code

Ansys nCode Design Life

Advanced

>> Duration

2 days

>> Participants

Mechanical Engineers, Design Engineers including Vibration, automotive, industrial equipment engineers
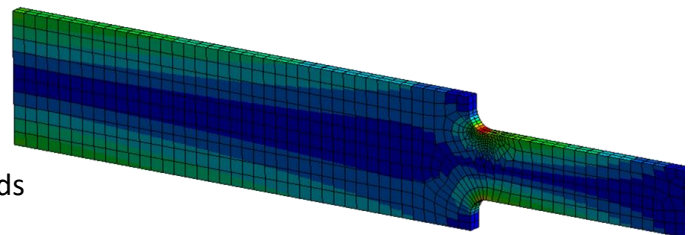
>> Prerequisites

Engineering knowledge fundamentals, experience with Ansys Mechanical and basic theoretical background in fatigue analysis
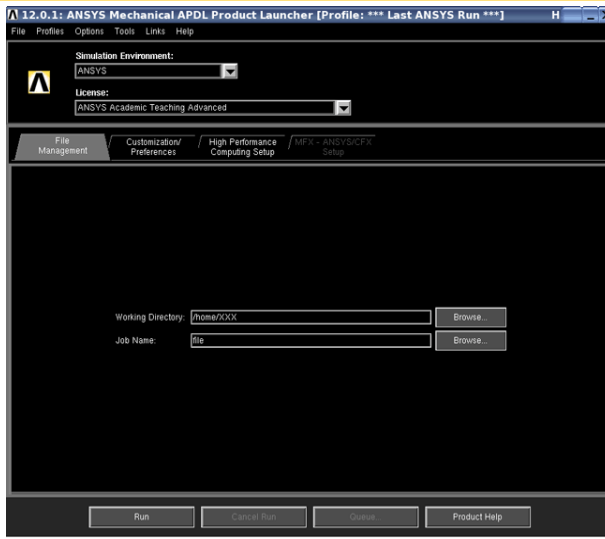
Overview

Participants in this advanced training course will learn how to utilize the Ansys nCode Design Life tool for stress-life and strain-life fatigue analysis on structures under fatigue loading. They will also acquire techniques for creating accurate models of complex fatigue scenarios through material mapping, load mapping, and duty cycle methods. Upon completion, participants will have the skills and knowledge to analyze fatigue in structures effectively with the Ansys nCode Design Life tool.

Topics

- Introduction to fatigue analysis
- SN fatigue analysis
- EN fatigue analysis
- Fatigue analysis with non-constant loads
- Vibration fatigue

Introduction to Ansys Mechanical APDL (Classic Environment)

4 days

>> **Participants**

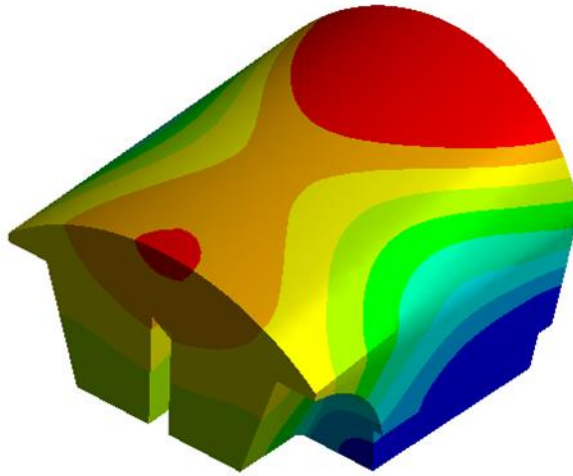Mechanical Engineers, Design Engineers

>> **Prerequisites**

Engineering knowledge fundamentals, experience in FEA

Overview

This course provides an introduction to Ansys Parametric Design Language (APDL), including the associated workflow for building, solving, and post-processing simulation models. Participants will learn how to work at the node and element levels of a finite element model, providing maximum control over its behavior. This course is designed to equip participants with the skills and knowledge necessary to effectively utilize APDL in their simulation workflow.

Topics

- Graphical user interface
- Software general parameters
- Import and geometry creation
- Element library
- Mesh generation
- Material definition
- Load definition
- Getting a solution and solver selection
- Static, thermal and modal analysis & parameters
- Constrain equations, coupling and contact creation

Acoustics Analysis in Ansys Mechanical

Advanced

>> Duration

2 days

>> Participants

Mechanical Engineers, Design Engineers
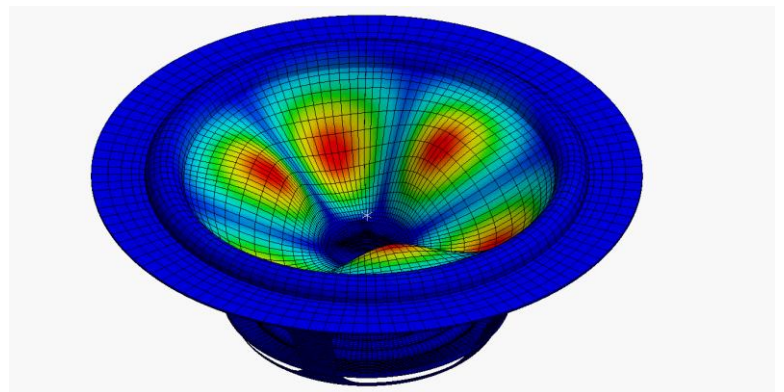
>> Prerequisites

Engineering knowledge fundamentals & experience with Ansys Mechanical

Overview

This course aims to equip learners with the necessary skills to use Ansys Mechanical for Acoustics analysis. Participants will gain a general understanding of acoustic phenomenon, terminology, and governing equations, while also learning the procedures for performing modal and harmonic acoustics analysis in Ansys Mechanical.

Topics

- Introduction to acoustics
- Modal analysis
- Harmonic analysis
- Transient analysis
- Advance applications
- Car acoustic models

Introduction Ansys ACT

>> **Duration**

2 days

>> **Participants**

Mechanical Engineers,
Design Engineers

>> **Prerequisites**

Engineering knowledge
fundamentals, experience with
Ansys Mechanical & basic
knowledge of Python
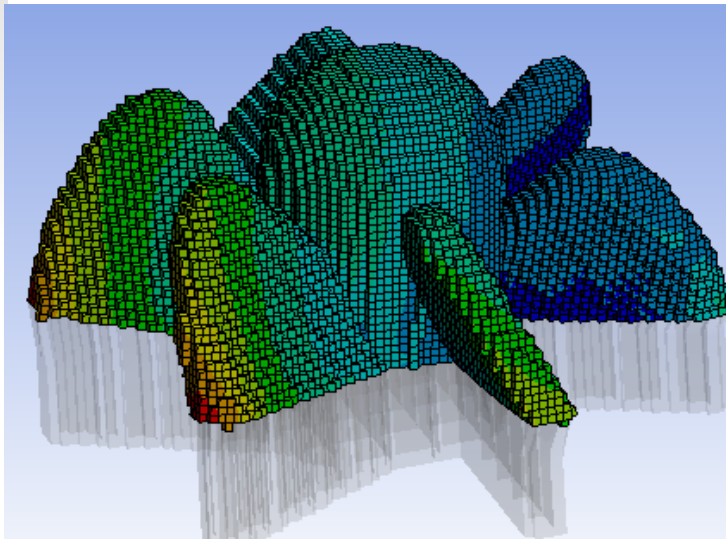programming is recommended

Overview

This training course focuses on the Ansys Application Customization Toolkit (ACT) in Mechanical Workbench. Participants will learn how to automate the creation of standard tree objects in Mechanical and its integrated modules, as well as how to create custom loads and results. The course also covers the capabilities of ACT, providing learners with a comprehensive understanding of its potential applications.

Topics

- Introduction to ACT
- Applications examples
- XML and Python programming
- Iron Python console
- Macros integration
- Development of customized results
- Advanced functionalities
- Practical cases

Introduction to Ansys Additives

Advanced

>> Duration

2 days

>> Participants

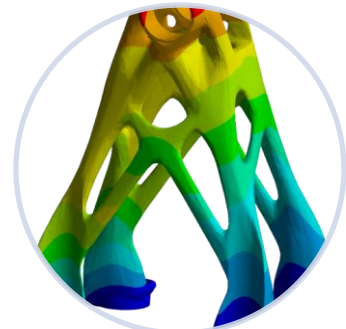Mechanical Engineers, Design Engineers

>> Prerequisites

Engineering knowledge fundamentals – FEA, Ansys Mechanical & Ansys SpaceClaim

Overview

This training course provides basic to intermediate level knowledge on 3D print process simulation. It covers the general overview of the challenges that arise when simulating a print process.

Topics

- Additive manufacturing simulations
- Calibration
- Additive wizard
- Inherent strain analysis
- Distortion compensation using SpaceClaim
- Inherent strain analysis with symmetry
- Thermomechanical simulation
- Heat treatment analysis
- Advanced settings
- DED process
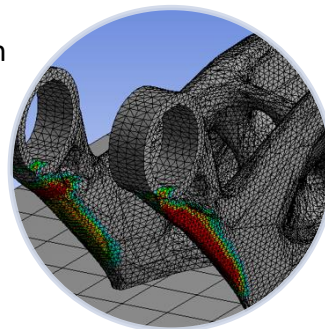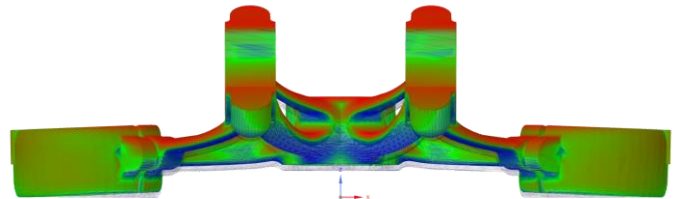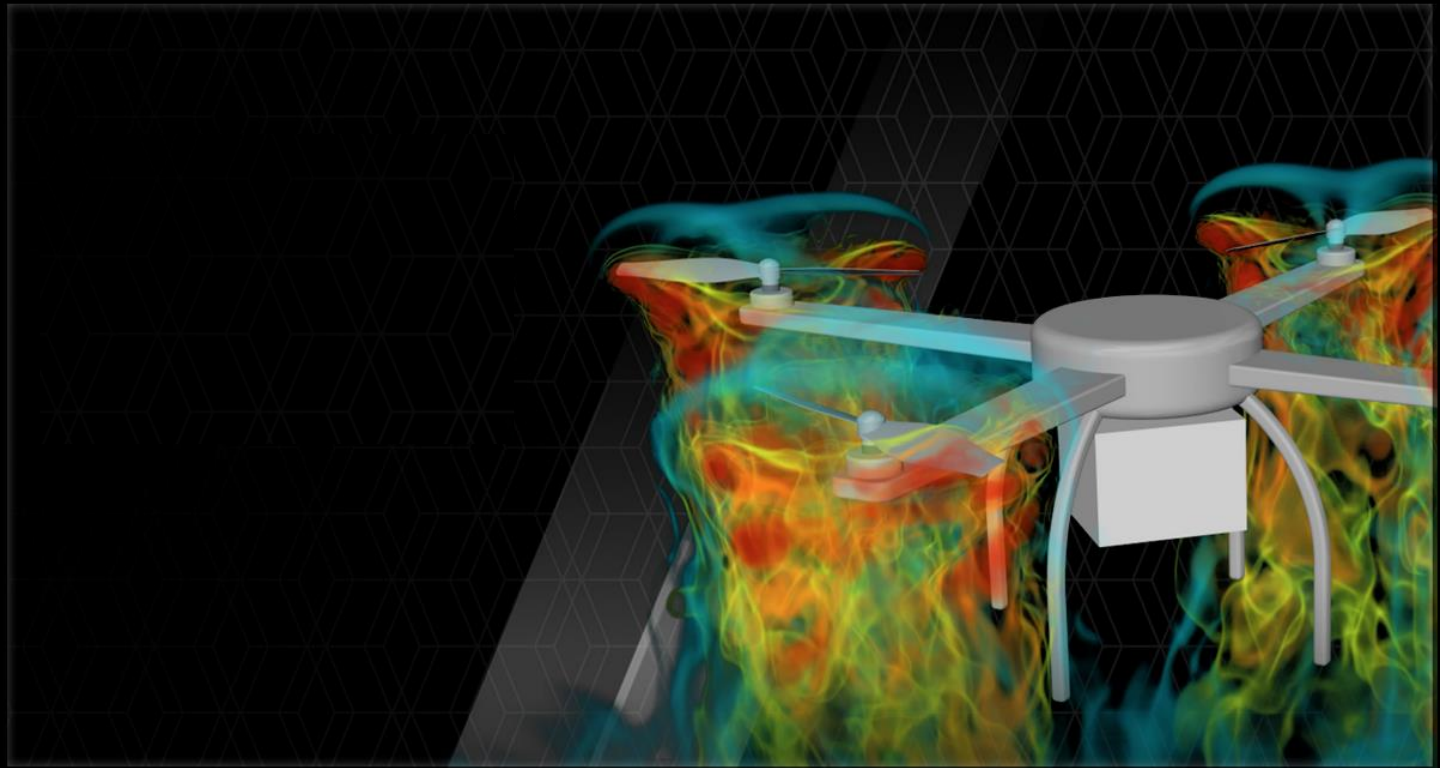
Fluids

External Aero-Thermal

Aircraft Systems

Hypersonic Flows

Icing

Propulsion

Airborne Noise

Introduction to Ansys Fluent

Standard

>> Duration

4 days

>> Participants

Engineers and Designers

>> Prerequisites

Engineering knowledge,
basics in fluids & heat transfer

Overview

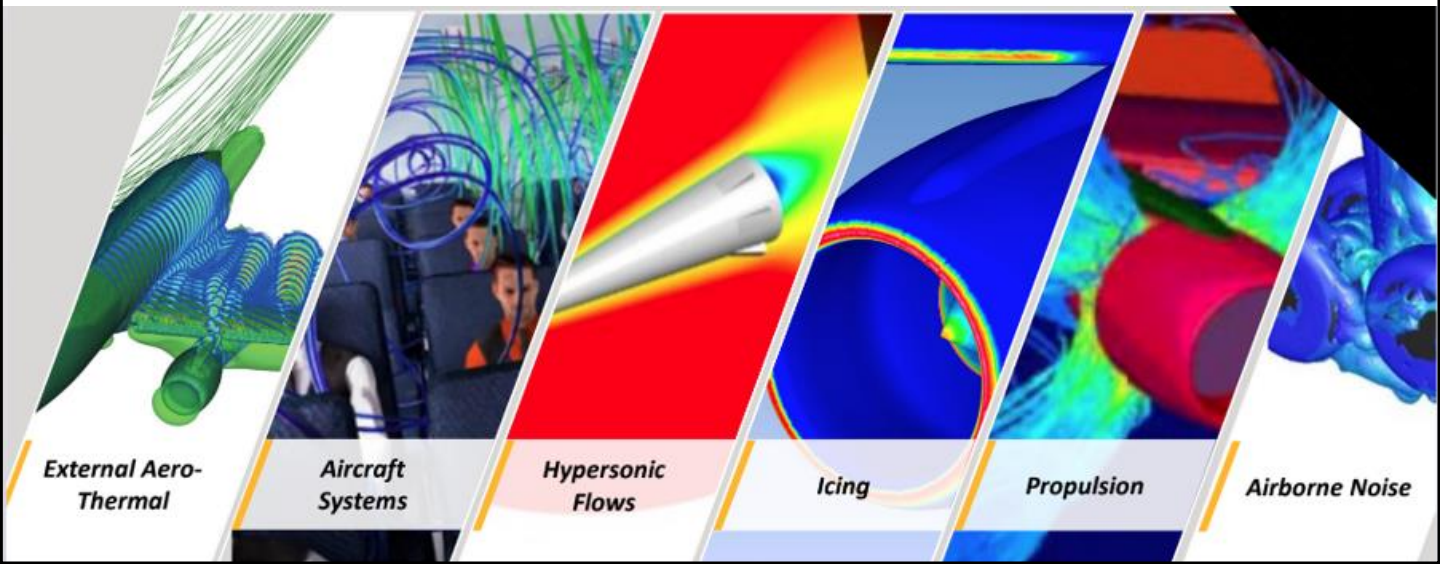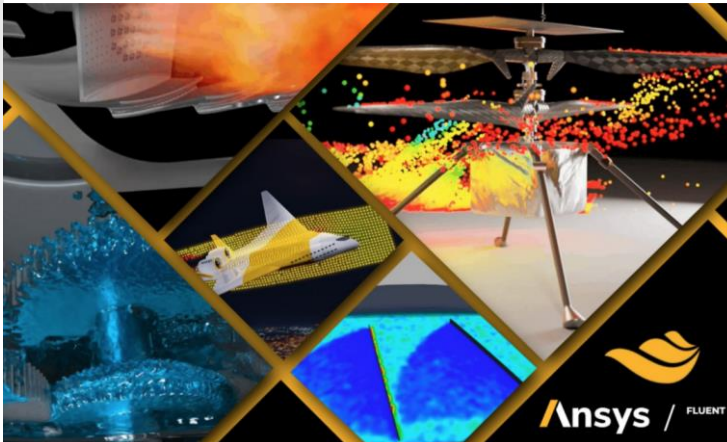The Ansys Fluent training program provides an in-depth understanding of Computational Fluid Dynamics (CFD) modeling, covering the complete simulation process from geometry preparation and mesh generation to post-processing results. Participants will learn best practices for conducting popular CFD simulations, advanced post-processing features, and gain skills to apply this knowledge to solve real-world fluid dynamics problems.

Topics

- Overview of CFD process
- Geometry preparation using SpaceClaim
- Meshing using Fluent meshing
- Solving using Ansys Fluent
- Results and postprocessing
- Parametrization and design points
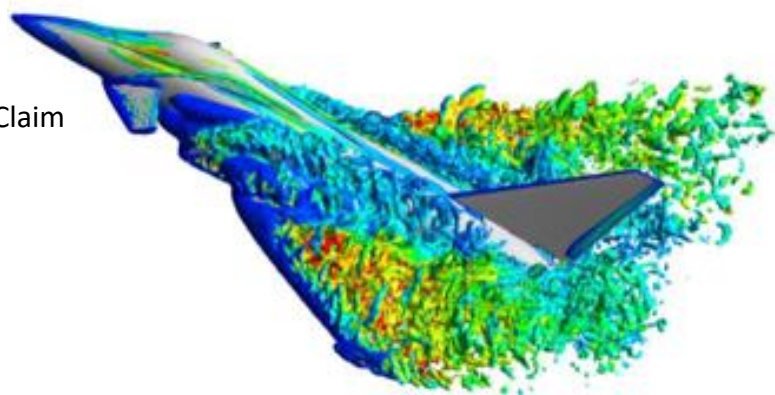
Introduction to Ansys CFX

>> Duration

4 days

>> Participants

Engineers and Designers

>> Prerequisites
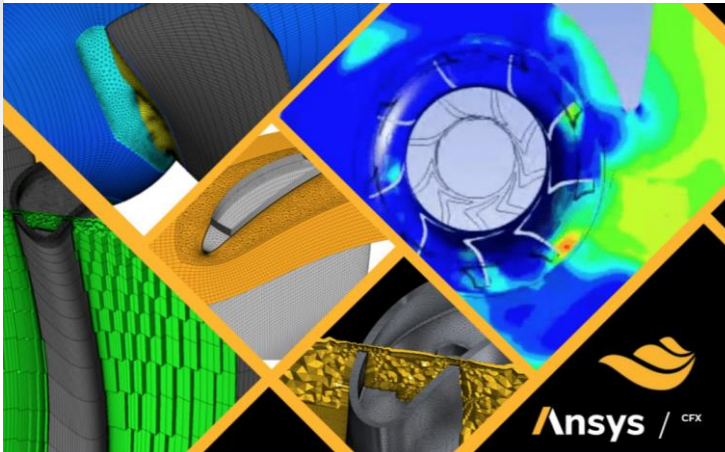
Engineering knowledge, basics in fluids & heat transfer

Overview

This training program focuses on providing participants with the fundamental skills needed to conduct basic computational fluid dynamics (CFD) simulations using Ansys CFX software. It covers the entire CFD workflow, from working with CAD models in Ansys Spaceclaim, to generating high-quality CFD meshes with Ansys Fluent Meshing, and finally, performing all aspects of CFD simulations in Ansys CFX. By the end of the training, participants will have a solid understanding of the CFD simulation process and be able to conduct basic CFD simulations using Ansys CFX software.

Topics

- Overview of CFD process
- Geometry preparation using SpaceClaim
- Meshing using Fluent Meshing
- Solving using Ansys CFX
- Results and postprocessing
- Parametrization and design points
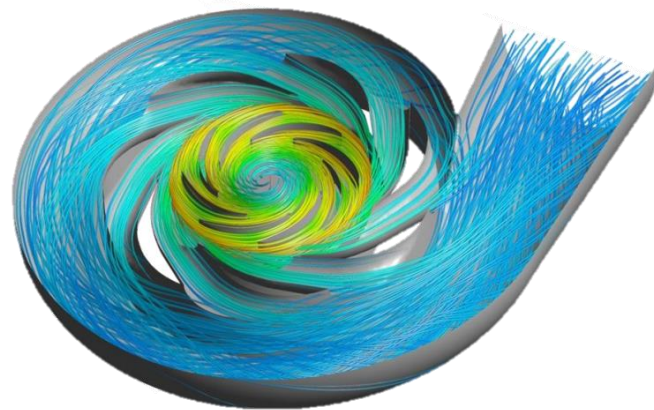
Ansys Chemkin-Pro & Model Fuel Library

>> Duration

4 days

>> Participants
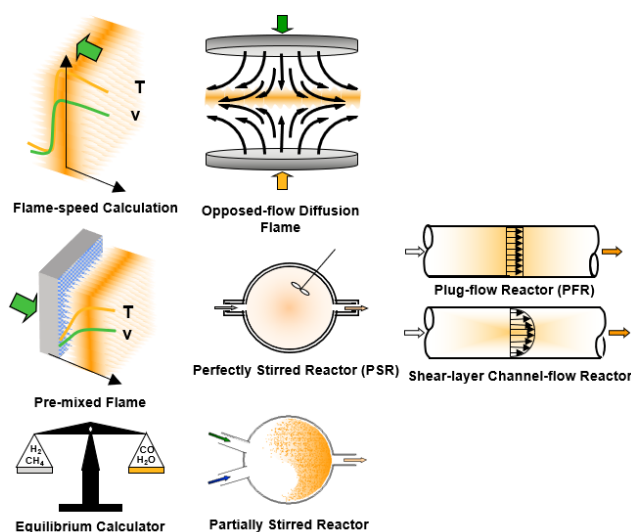
Engineers and Designers

>> Prerequisites

Technical education and/or background in fluids

Overview

Ansys Chemkin-Pro offers powerful tools for simulating detailed chemistry that is crucial for designing systems with reduced pollutant emissions and undesired byproducts in a cost-effective manner. This course is designed to provide participants with hands-on experience using Ansys Chemkin-Pro to select kinetics models, perform simulations of different reactor types, analyze complex systems, understand critical reactions, and ensure the accuracy of chemistry models used in CFD. Upon completion of the program, participants will be proficient in using Ansys Chemkin-Pro for simulating detailed chemistry and receive a certificate of completion.

Topics

- Introduction and chemistry fundamentals
- Combustion and emission modeling
- Catalysis and materials
- Multiphase processes framework

Ansys Polyflow is now Mat-Pro
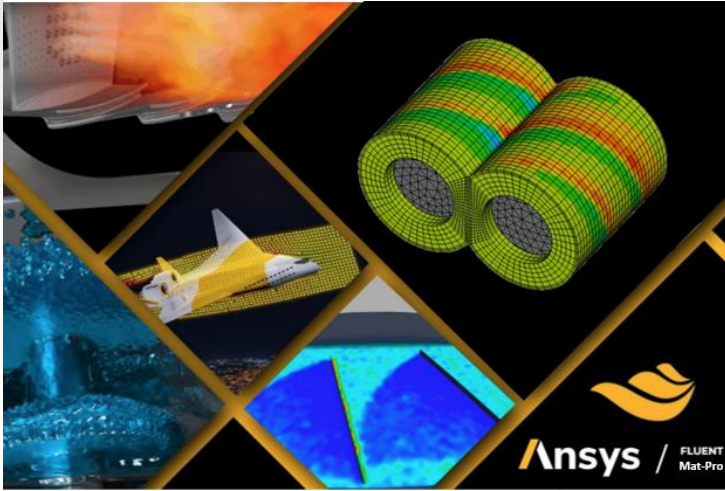
>> Duration

2 days

>> Participants

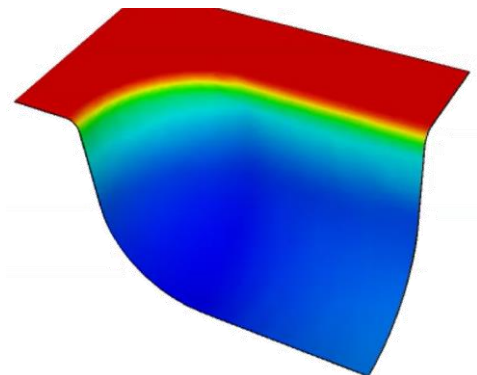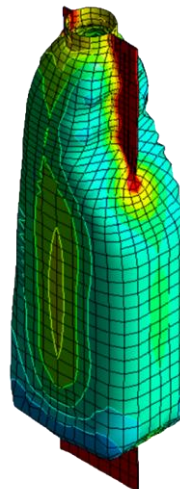Engineers and Designers

>> Prerequisites

Technical background in fluids, pre-processing using SpaceClaim, Ansys/Fluent Meshing
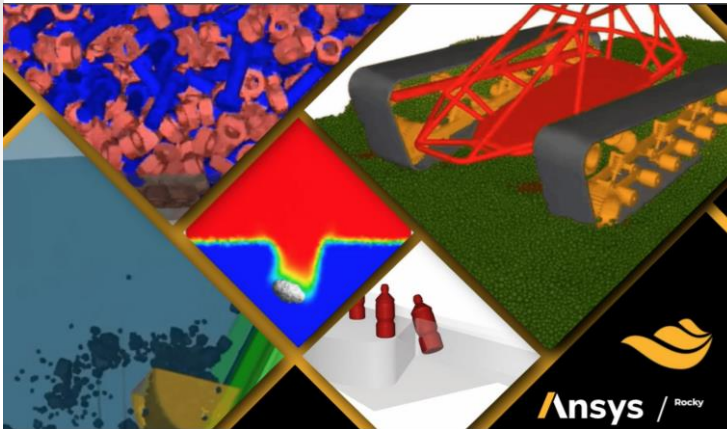
Overview

The Ansys Polyflow training course focuses on manufacturing applications, teaching participants how to effectively use the software to simulate complex flows, identify potential manufacturing issues, and optimize product quality and production costs. Through this training, participants can improve their work efficiency and enhance their team's skills, leading to better manufacturing practices and increased profitability. The course also covers the Fluent Materials Processing Workspace, providing a comprehensive understanding of the latest manufacturing tools to solve real-world problems.

Topics

- Ansys Polyflow overview
- 2.5D extrusion
- 3D extrusion
- Modeling forming
- Time dependent flows
- Workbench parametrization
- Post processing using CFD-Post

Ansys Rocky

>> Duration

3 Days

>> Participants

Engineers and Designers
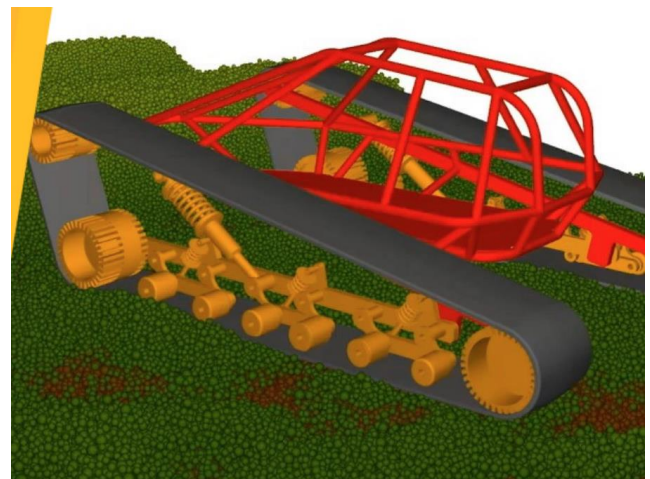
>> Prerequisites

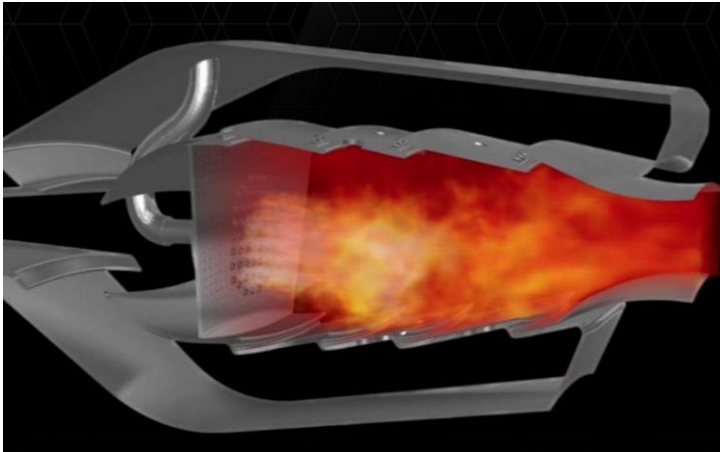Engineering knowledge in fluids

Overview

The Ansys Rocky training offers participants the opportunity to gain a comprehensive understanding of the software's capabilities and apply them to real-world problems. By attending this training, customers can learn how to optimize their processes, reduce costs, and increase efficiency by accurately modeling and analyzing particle behavior. Additionally, they will learn how to extract insights and predictions that can improve the design and operation of equipment, saving valuable resources and time.

Topics

- Introduction to DEM and basic concepts
- Particle shapes and motion frames
- Post-processing in Rocky DEM
- Additional contact force models and coarse grain model

Ansys Fluids

>> Duration

1 day / 2 days

>> Participants

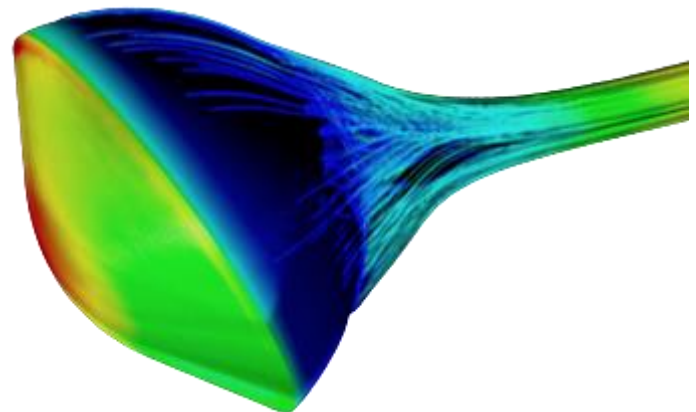Engineers and Designers

>> Prerequisites

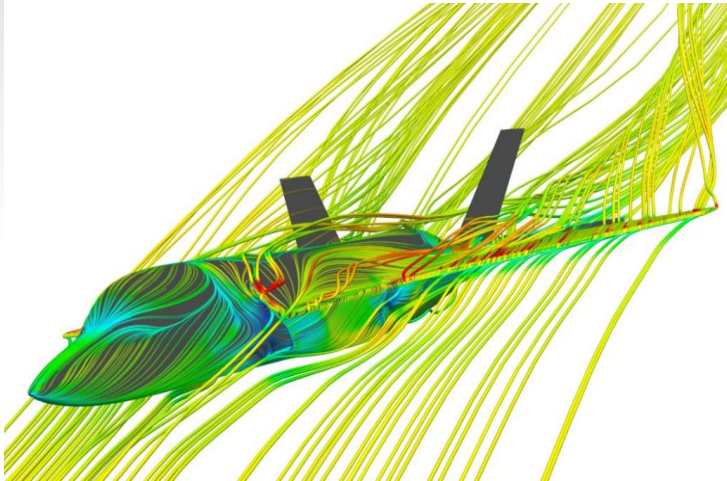Engineering knowledge in fluids, Ansys Fluent, Ansys CFX

Overview

This advanced training is tailored for Ansys Fluent and CFX users with subject matter expertise in combustion, multiphase, turbulence, and more. It aims to provide the latest industry-specific knowledge and enhance participants' skills. The training covers various topics, including aeroacoustics behavior prediction, combustion process modeling, complex multiphase flow simulation, turbulence effects prediction, and turbomachinery aero-thermodynamics. By completing this training, participants can improve their expertise in CFD simulations and deliver more accurate and efficient solutions to their industry-specific challenges.

Topics

- Aeroacoustics modeling
- Combustion modeling
- Multiphase modeling
- Turbulence modeling
- Turbomachinery modeling
- Fluid structure interaction FSI modeling
- Advanced DPM - spray modeling

External Aerodynamics

Advanced

>> Duration

1 day

>> Participants
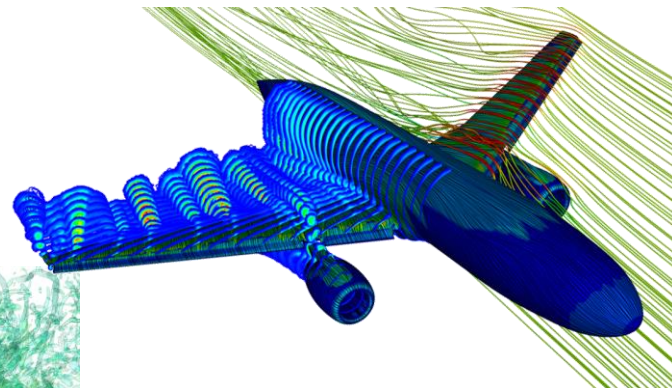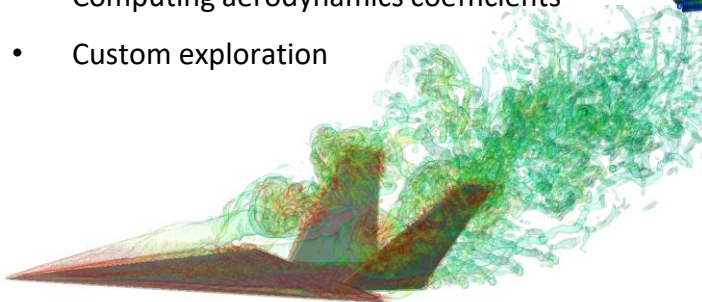
Engineers and Designers

>> Prerequisites

Engineering knowledge in fluids, Ansys Fluent, aerodynamics

Overview

Ansys Fluent Aero Workspace is a powerful tool for aerospace and defense engineers, providing customized workflows to access automation enhancements, High-Speed Numeric (HSN), and convergence best practices. This training is designed to cover all aspects of the Fluent Aero Workspace, from creating a project to setting up parametric simulations. Participants will gain a deep understanding of the Fluent Aero Workspace, allowing them to efficiently and effectively tackle complex aerodynamic problems. This training will help aerospace and defense engineers to optimize designs, reduce costs, and improve the overall performance of their products.
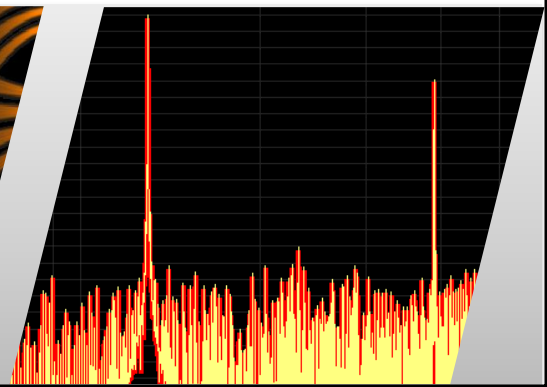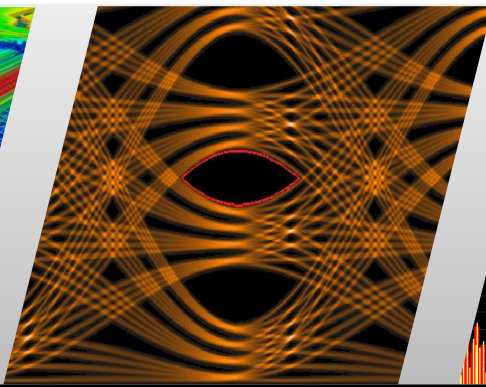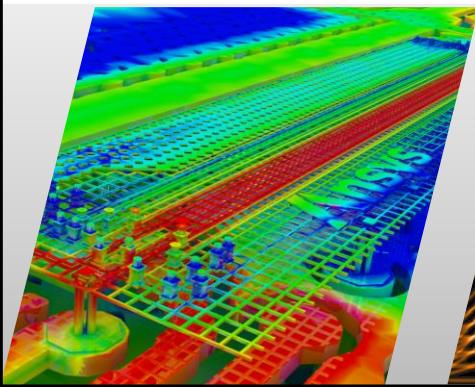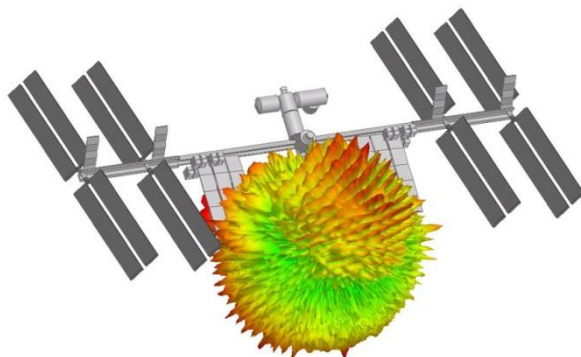
Topics

- Fluent Aero overview
- Parametric setup
- Viewing the results inside Fluent Aero
- Computing aerodynamics coefficients
- Custom exploration

Electromagnetics

Introduction to Ansys HFSS
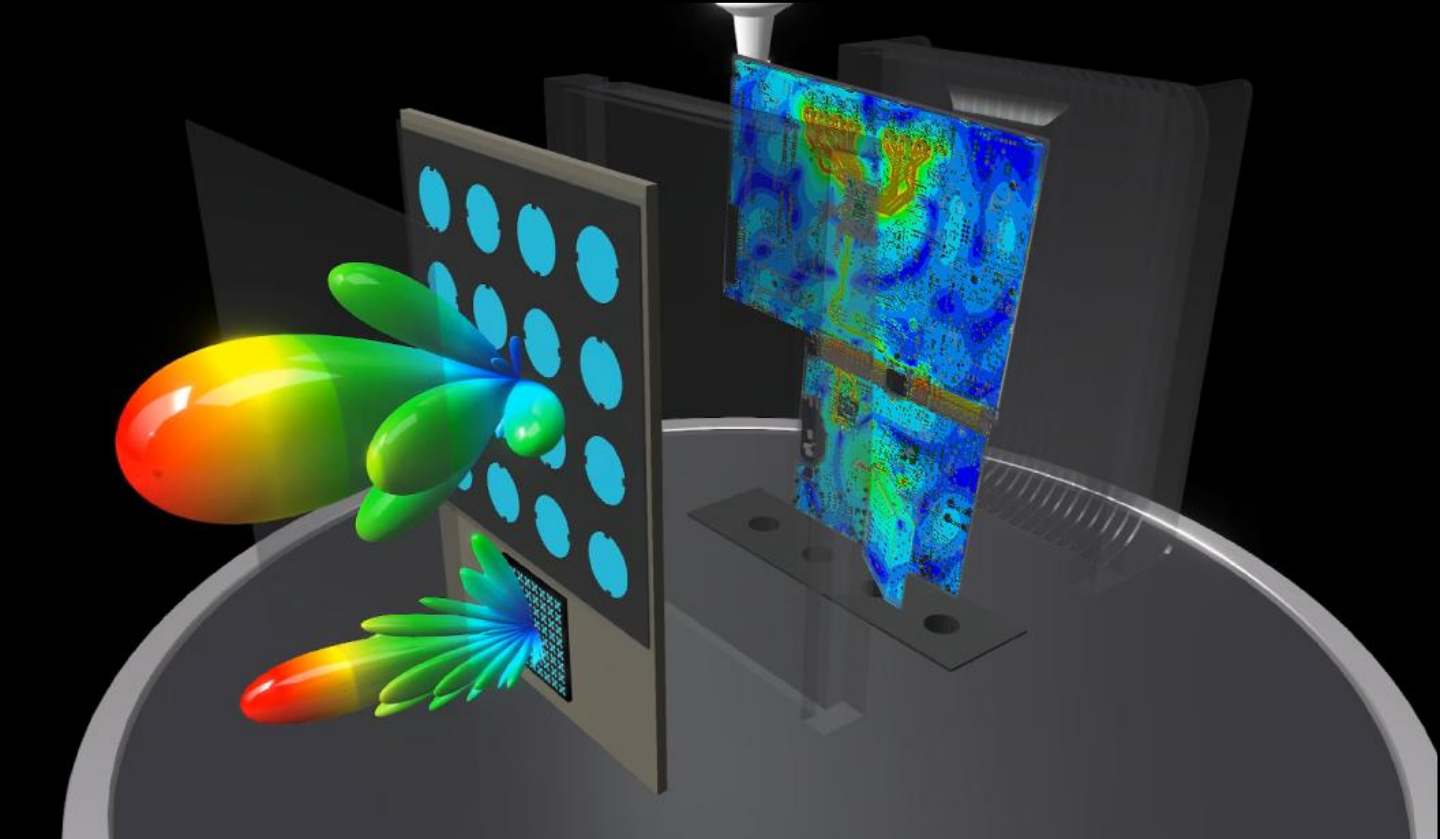
>> Duration

2 days

>> Participants

Engineers and Designers (HF)

>> Prerequisites

Technical education and/or background in high frequency electromagnetics

Overview

This is an introductory to intermediate-level training program for using Ansys HFSS in all applications, such as RF/microwave, antennas, or planar problems. Participants will gain an understanding of HFSS modeling, solution processing, and post-processing features that can be applied to other advanced applications. The course will also cover advanced topics, such as the dynamic link between EM and circuit, impedance matching, an overview of the HFSS 3D layout interface, and speeding up HFSS simulation using HPC.

Topics

- HFSS overview
- Boundary conditions and excitations
- Setup and solution options
- Meshing options
- HPCS and its setup
- Post-processing options
- Dynamic link between EM and Circuit
- HFSS 3D layout overview

Introduction to Ansys SIwave

Standard
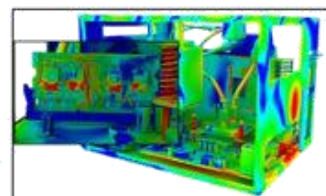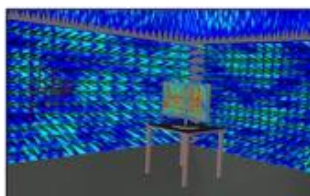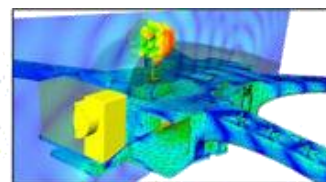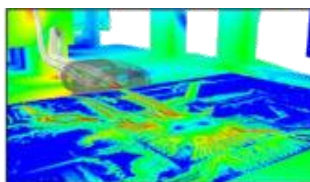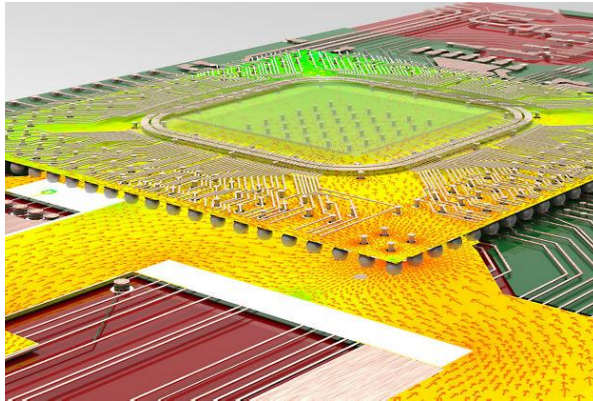
>> Duration

2 days

>> Participants

Electronics engineers involved in high speed PCB design

>> Prerequisites

Familiarity with PCB layouts
Familiarity with high-speed digital signal electronic engineering

Overview

SIwave is an advanced analysis and design tool for complex PCBs, packages, silicon interposers, and RDLs. By employing multiple state-of-the-art full-wave EM solvers, SIwave helps designers solve SI, PI, and EMI/EMC problems of chip/package/board systems. In addition to generating S-parameters, RLCG extractions, and SPICE netlists, SIwave offers a variety of analyses including impedance scanning, DC-IR drop, time-domain reflectometry (TDR), and impedance optimization of PDN using decoupling capacitors. Ports, terminations, and circuit elements can be inserted into the design to set up the simulation and model the system end-to-end.

Topics

- SIwave overview
- Import and scans
- Signal integrity
- Power integrity
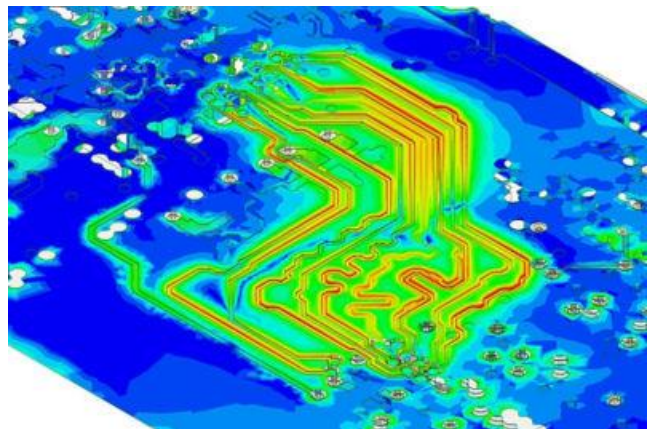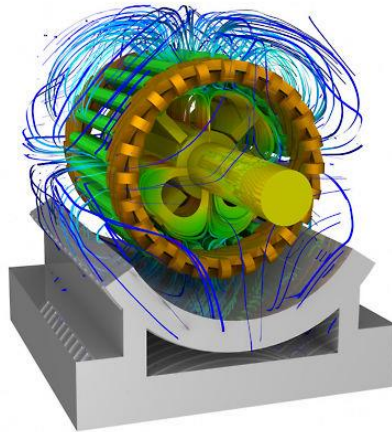
Introduction to Ansys Maxwell

Overview

The Ansys Maxwell standard course introduces learners to the static and quasistatic solvers available in Ansys Maxwell, which operates in the Ansys Electronic Desktop (AEDT). Electric solvers include Electrostatic, and Electric Transient. Magnetic solvers include Magnetostatic, Eddy Current and Magnetic Transient. Most workshops include geometry construction instructions, but the workshops also provide files for learners not interested in 3D modeling practice.

Topics

- Maxwell overview
- Introduction to the finite element method
- Electrostatics
- DC conduction
- Magnetostatics
- Parametric and Optimetrics
- Transients simulations
- Post-processing

Introduction to Ansys Icepak

Standard
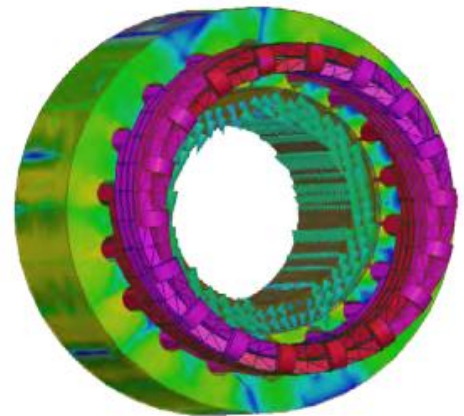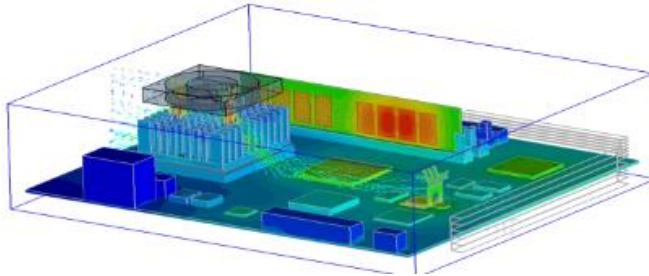
>> Duration

2 days

>> Participants

Thermal, Mechanical and Electrical engineers

>> Prerequisites

Technical education in electronics and/or fluid mechanics and heat transfer

Overview

Ansys Icepak provides flow and thermal management solutions for many types of electronic design applications. The primary goal of this course is to cover the basics of using Ansys Icepak in the Ansys Electronics DeskTop (AEDT) user environment. users will be introduced to the world of electronics thermal modeling through a combination of lectures, workshops and examples/demonstrations.

Topics

- Icepak in AEDT Overview
- Interface and modeling basics
- Meshing
- Solution setup and post-processing
- Electro-thermal analysts
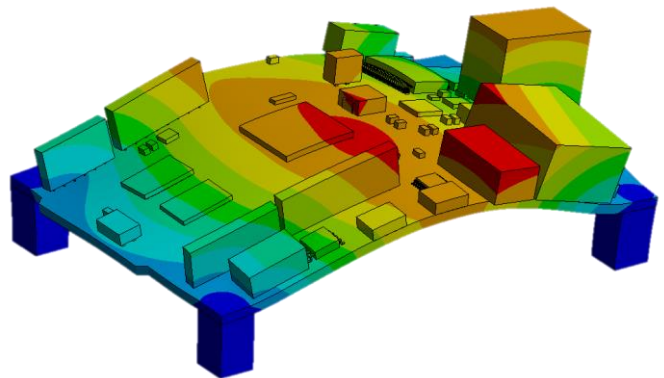
Antenna Array and Periodic Structures
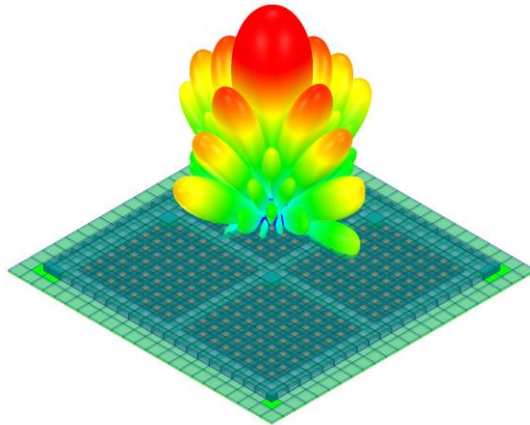
Advanced

>> **Duration**

1 days

>> **Participants**

Engineers and Designers

>> **Prerequisites**

Technical education and/or background in high frequency electromagnetics

Overview

The HFSS Antenna Arrays and Periodic Structures course explores topics such as Infinite Array (Unit Cell analysis), Finite Array with both periodic and semi-periodic (non-identical) unit cell structures, Scanning the beams using Finite Array Beam Angle Calculator Toolkit, and Frequency Selective Surfaces (FSS). This course is designed for intermediate to advanced users and includes four modules with workshops that demonstrate the workflows from start to finish.

Topics

- Unit cell
- Finite array domain decomposition method
- Component array domain decomposition method
- Frequency selective surface

Ansys HFSS SBR+ Radar Cross Section (RCS)

Advanced

>> Duration

1 days

>> Participants

Engineers and Designers

>> Prerequisites

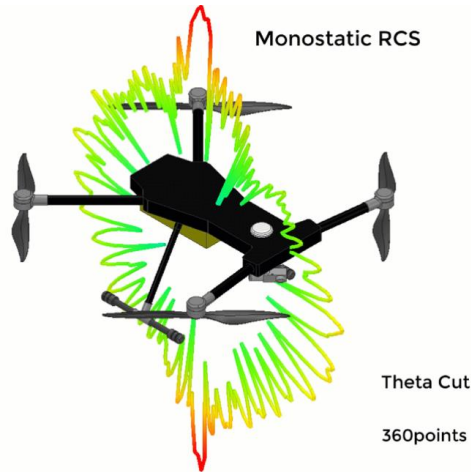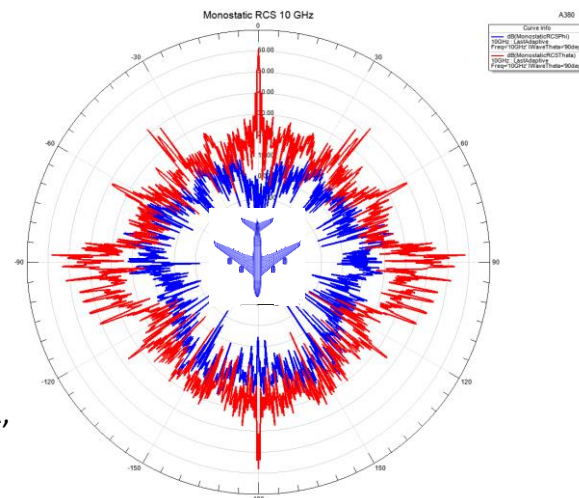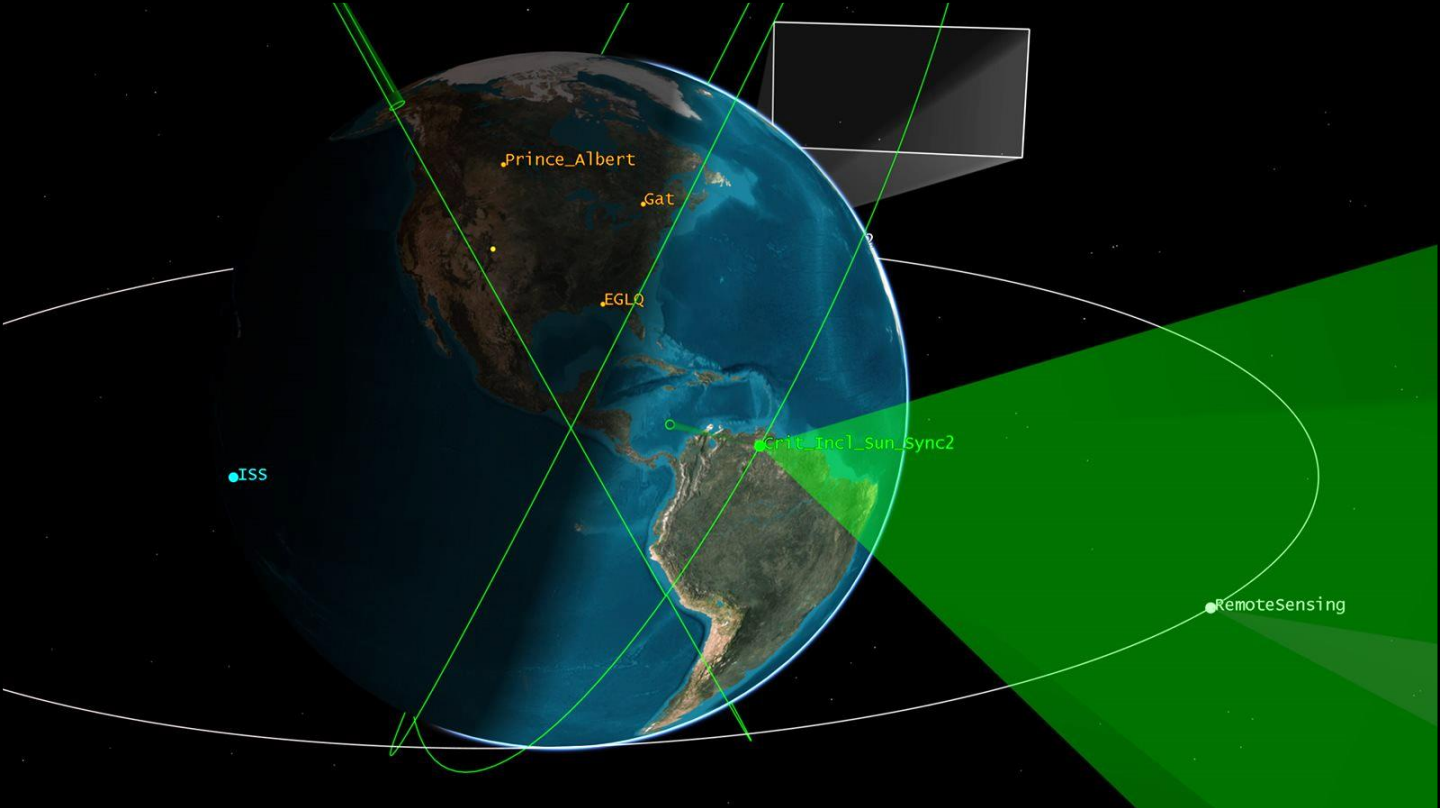Technical education and/or background in high frequency electromagnetics

Overview

The Ansys HFSS SBR+ RCS course provides a comprehensive understanding of radar cross-section (RCS) applications using the shooting bouncing rays (SBR) formulation. The course covers both monostatic and bistatic RCS, as well as radar signature imaging applications such as range profile, waterfall plots, and ISAR. It also includes technical descriptions of PTD (physical theory of diffraction), related wedge settings, the development of SBR+ surface currents on scattering geometry, and UTD (uniform theory of diffraction). Additionally, the course provides a detailed explanation of HFSS ACT Extension utilities RADARpre and RADARpost. This course is designed for experienced radar signature engineers who are familiar with Ansys HFSS for antenna applications..
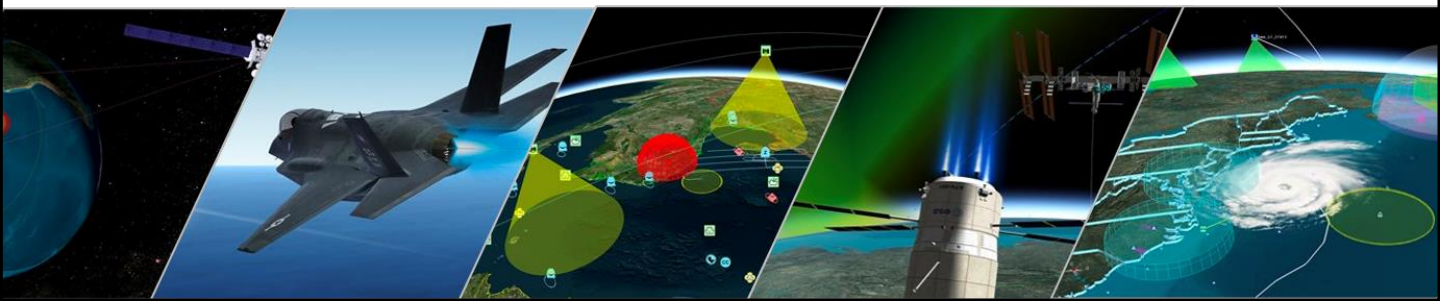
Topics

- RCS using the SBR+ design type
- Plot 2D radiation pattern simulation results
- Simulate and plot 3D monostatic RCS "fuzzballs"
- Compare SBR+ formulation with IE formulation
- Use ACT extensions to set up range profiles, ISAR, and waterfall plots.

Digital Mission Engineering

Introduction to Ansys STK

>> Duration

3 days

>> Participants

Aerospace Engineers, Antenna Design Engineer, Satellite Operators, System Engineers and Thermal Engineers.

>> Prerequisites
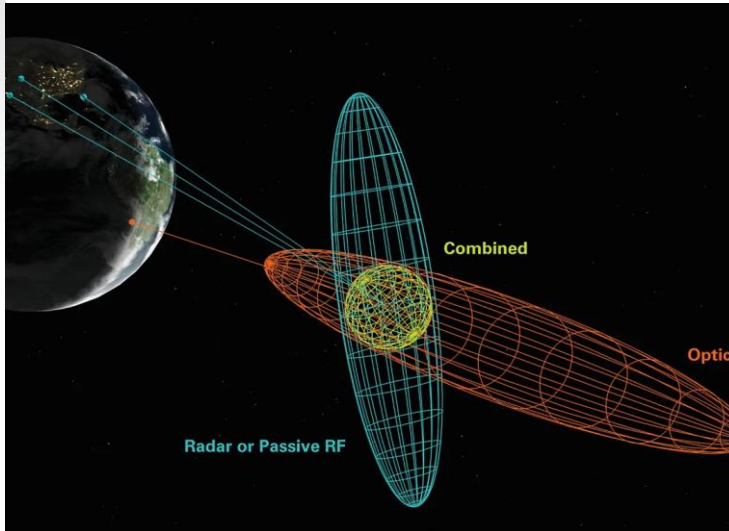
Technical education and/or background in aerospace.

Overview

Ansys Systems Tool Kit (STK) provides a physics-based modeling environment for analyzing platforms and payloads in a realistic mission context. The primary goal of this course is to familiarize students with the STK workflow then build up to the advanced analysis capabilities and tools to quantify and measure mission effectiveness.

Topics

- Building scenarios
- Coverage and volumetrics
- Trade studies with Analyzer
- Modeling aircraft missions
- Designing spacecraft trajectories
- Communication & radar analysis
- Integration and automation
- EOIR sensors

Introduction to Ansys ODTK

>> Duration

3 days

>> Participants

Aerospace Engineers, Satellite Operators, System Engineers.
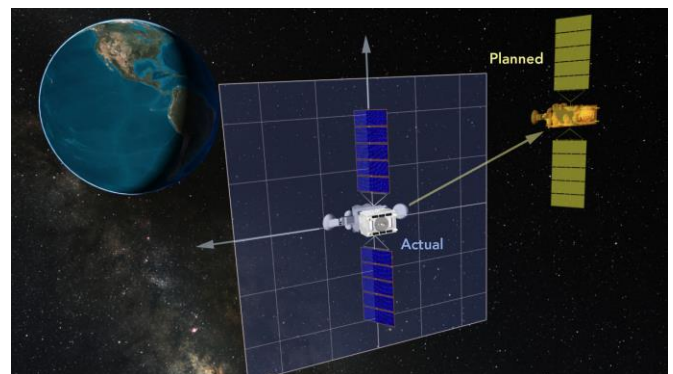
>> Prerequisites

Technical education and/or background in aerospace and orbit determination.

Overview

Ansys Orbit Determination Tool Kit (ODTK) provides highly accurate orbit estimates throughout the engineering life cycle, from mission design through operations. The primary goal of this course is to familiarize students with the orbit determination process and data processing of the associated results.

Topics

- Introduction to orbit determination
- ODTK orbit determination process
- Processing maneuvers
- Tracking system design

Customized Training

>> **Duration**

Customized

>> **Prerequisites**

Engineering Knowledge

Overview

When you need to go beyond what is offered in standard training or technical support for your specific Ansys product, consider Fluid Codes customized training. This solution is tailored to your own needs and enables you to spend one-on-one time with an experienced engineer looking in detail at your own application and its specific challenges, helping you with the usage of your Ansys product.

Contact us

sales@fluidcodes.com

www.fluidcodes.com

FLUID CODES

Ansys / ELITE CHANNEL PARTNER

UAE	**Saudi Arabia**	**Bulgaria**	**Egypt**
Mazaya Business Avenue BB1, Office 408. Jumeirah Lake Towers Dubai, UAE	Office number 126 Regus Novotel Business Park Dammam 31413, KSA	Office 807, 8th Floor 205 Alexander Stamboliyski blvd Sofia, Bulgaria	Sara's Tower-23, 3rd Floor, Batrawy street Nasr city Cairo, Egypt
Tel: +971 (0) 4 3308666	Tel: +966 13 845 2603	Tel: +359 877 43 59 58	Tel: +2 2 2617124