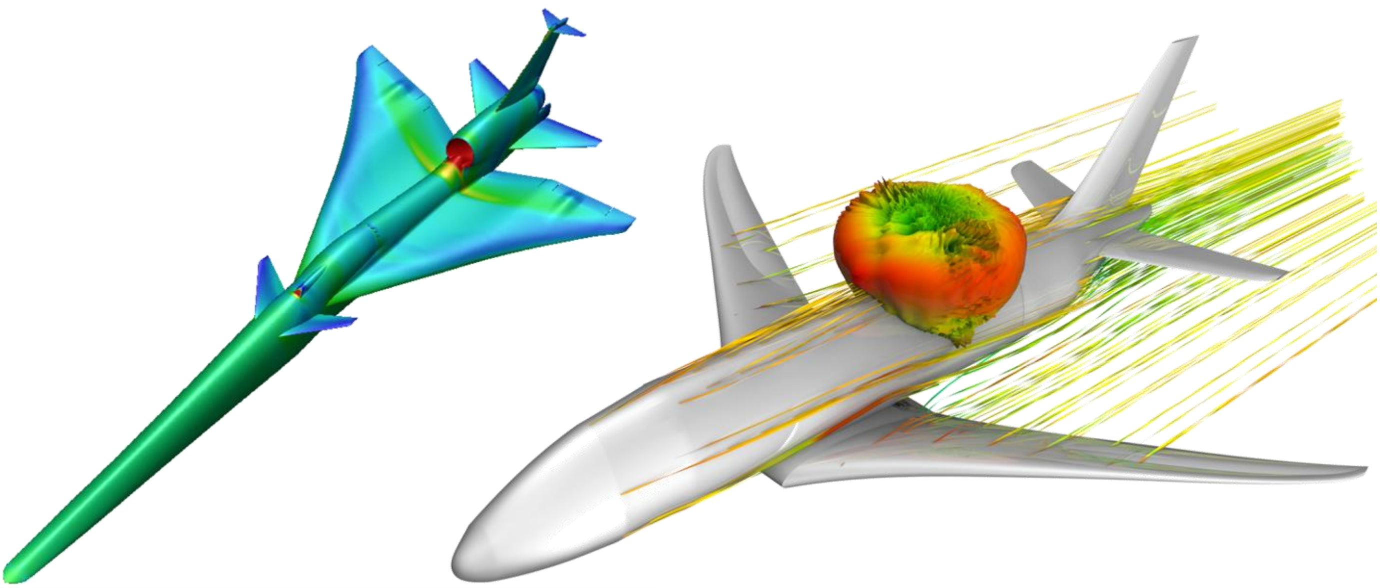
Training Catalogue



FLUID CODES

Ansys / ELITE CHANNEL PARTNER

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 made ANSYS the leader in the engineering simulation across EMEA region. Fluid Codes offers a wide range of training courses for the entire ANSYS product range, from beginners 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 from basic to the most advanced, to achieve the simulation goals. Through our standard on site training courses, our solutions enhances the participant to be successful in engineering using Ansys.

How can I request for a training?

Visit www.fluidcodes.com and go to the “Request to a Training” section or send us am email at training@fluidcodes.com.

Table of Contents

Training Information

What type of training are we offering?

- **Standard Training:** focuses on either, an introductory ANSYS knowledge or a physics specialization
- **Advanced Training:** focuses on advanced topics and their applications.
- **Customized Training:** focuses and adapted to specific application and simulation needs

Where can our training sessions take place?

- **In our offices**. In this case we provide the complete infrastructure for developing the training.
- **On Customer Site.** A Fluid Codes engineer will be sent to your company during the training period. We provide the temporary license keys and the computers are provided by either, the customer, or by us.

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

Some of our customer references

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.

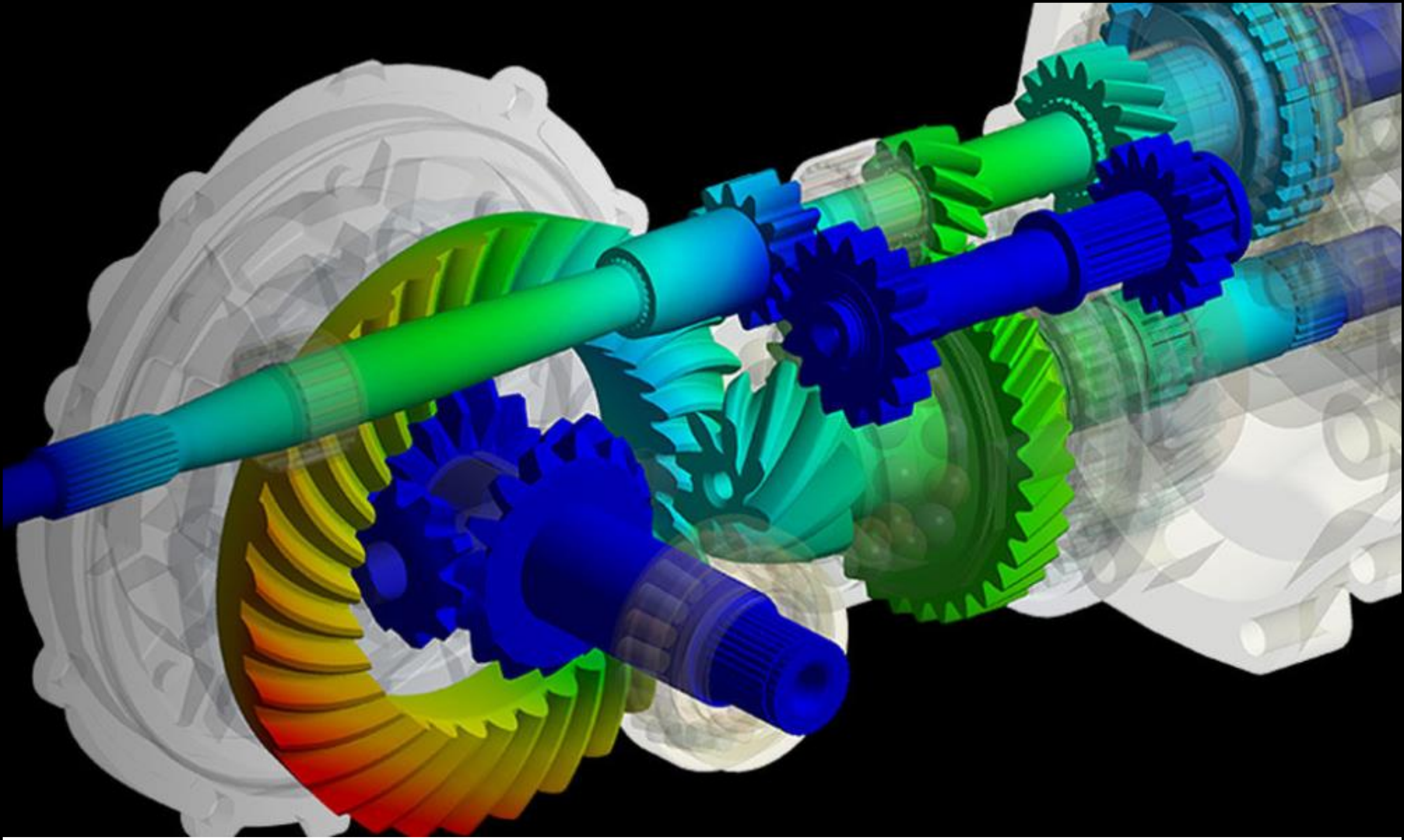
EXTERRAN BELLELI ENERGY
MASSIMILIANO CENTI
HEAD OF ENGINEERING

MASSIMILIANO CENTI
Head of Engineering

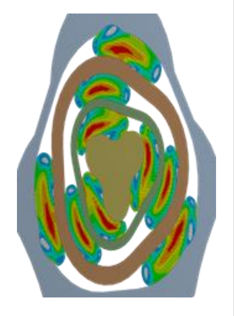
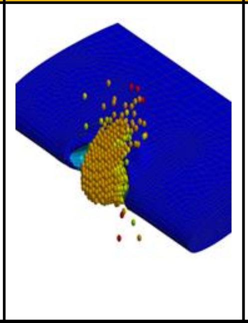
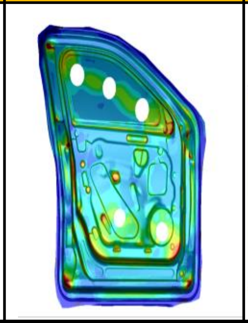
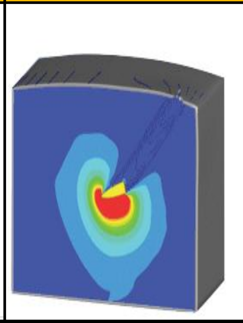
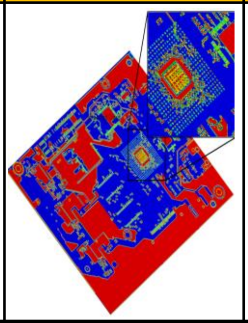
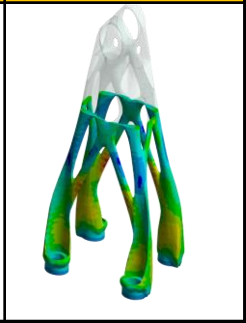
dar al-handasah
shair and partners

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

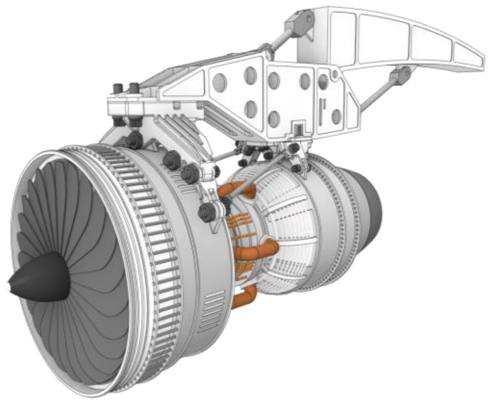


Structures

Mechanical	LS-DYNA	Forming	Auto-dyn	Sherlock	Additives
					

Geometry Preparation with ANSYS

Standard



>> Duration

1 day

>> Participants

Engineers and Designers

>>Prerequisites

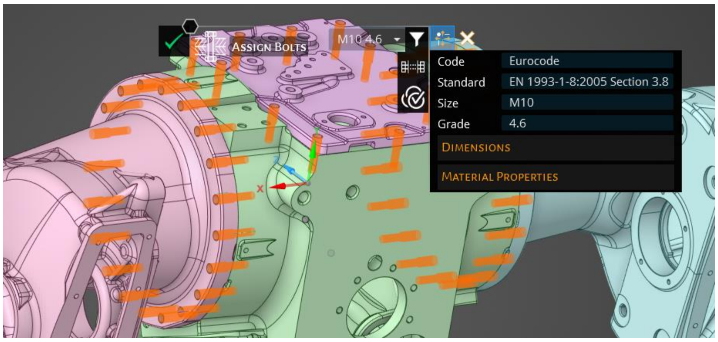
Engineering Knowledge

Overview

The Geometry preparation training course is for users who want to prep a CAD model for simulation in Ansys expert tools live, Mechanical, Fluent, Workbench etc. Topics covered include getting started content on the key geometry tools as well as complete workflows to prep a CAD model for structural and fluid flow simulations.. The training will be focused in either ANSYS Spaceclaim or ANSYS Discovery

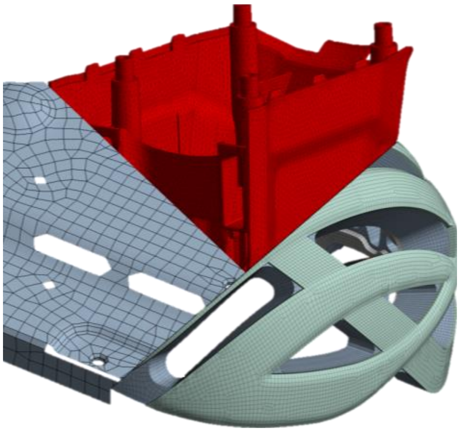
Topics

- 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



Standard

ANSYS Meshing for FEA



>> Duration

1 day

>> Participants

Engineers and Designers

>>Prerequisites

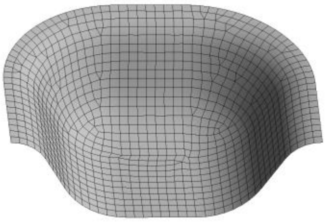
Technical education, and a background in fundamentals of FEA are recommended

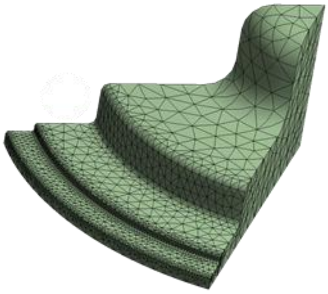
Overview

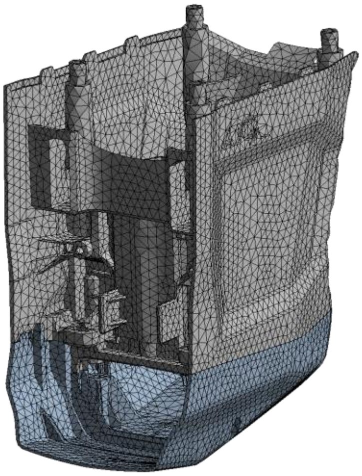
Ansys meshing for FEA training will present the principles of mesh generation, the workflow and best practices for efficiently creating meshes for Ansys FEA simulations.

Topics

- Mechanical Meshing core skills
- Meshing methods
- Global mesh controls
- Local mesh controls
- Mesh Quality and Advanced Topics

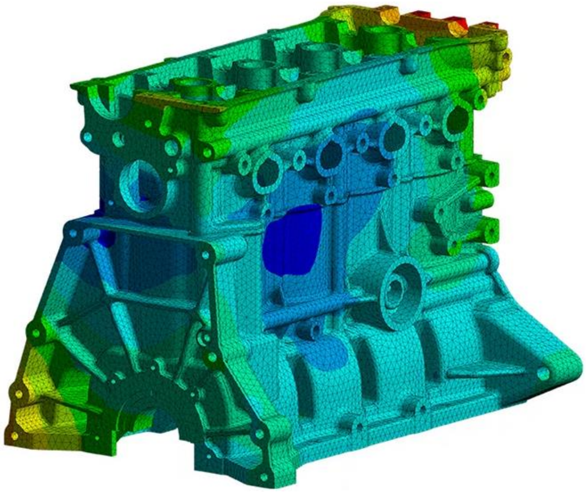






Standard

Introduction to ANSYS Mechanical



>> Duration

2 days

>> Participants

Engineers and Designers

>>Prerequisites

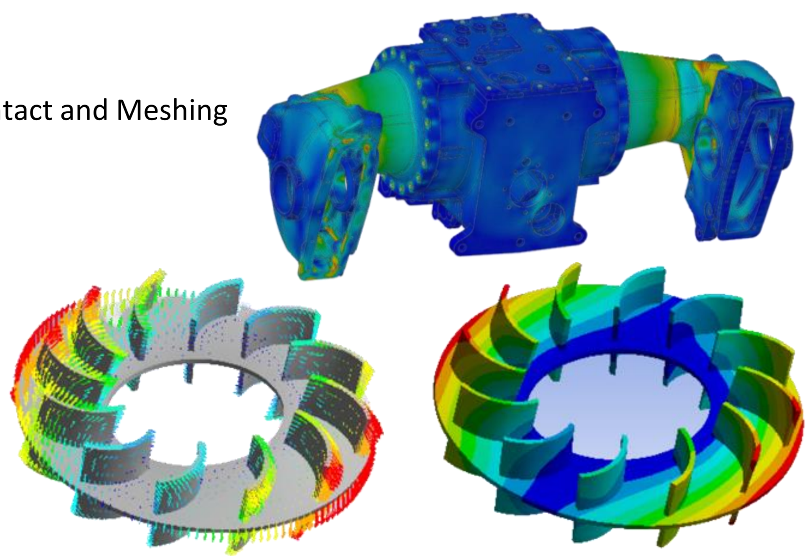
Engineering Knowledge
Fundamentals – FEA

Overview

In this course, participants will have the opportunity to start their simulation learning in Finite Element Structural Analysis (FEA) with the use of Ansys Mechanical. Finite Element Structural Analysis (FEA) allows to study the behavior of components or assemblies subject to operating conditions and flat loads for these structures.

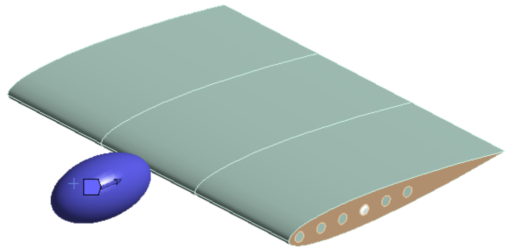
Topics

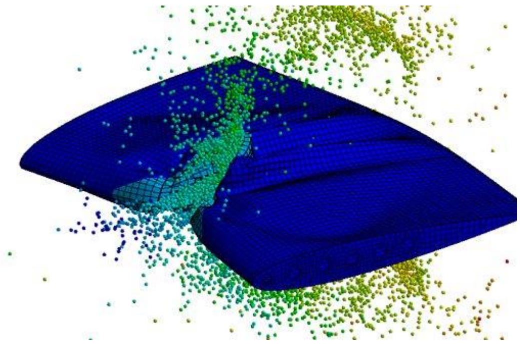
- General Pre-Processing, Contact and Meshing
- Static Structural
- Modal Analysis
- Steady State Thermal
- Post Processing
- Rigid Bodies
- Constraint Equations
- Multistep Analysis



Advanced

ANSYS Explicit Dynamics





>> Duration

2 days

>> Participants

Mechanical Engineers,
Impact Dynamics Engineers,
Drop Test Engineers

>>Prerequisites

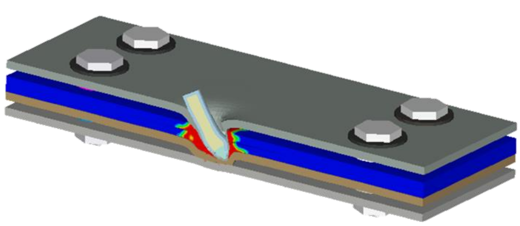
Engineering Knowledge
Fundamentals – FEA & Dynamics
ANSYS Mechanical

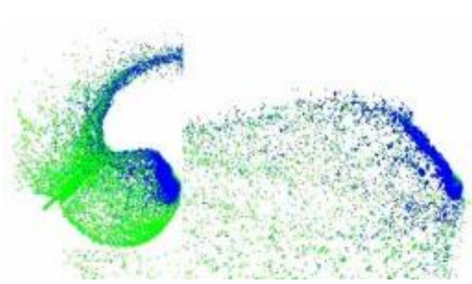
Overview

This training is addressed to design and analysis engineers who want to simulate high velocity dynamic problems, such as blasts, explosions, impacts. This is an advanced course to understand the difference between the explicit solution method to other methods used to perform dynamic analysis in Ansys and to choose the most appropriate, apply the proper explicit dynamic setup, proper meshing and apply the proper solution settings

Topics

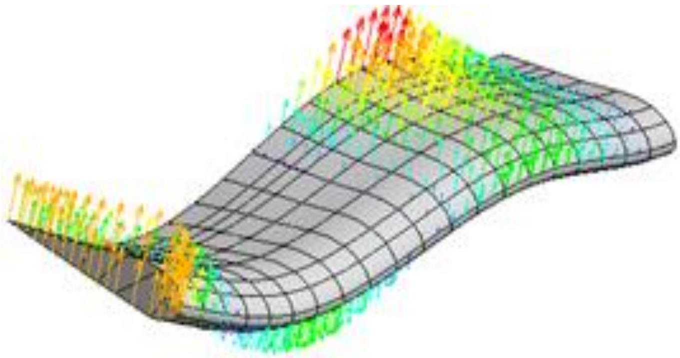
- Explicit Dynamics setup and workflow
- Introduction to Explicit
- Meshing in explicit dynamics
- Material Models
- Connections
- Analysis Settings configuration
- Euler and Particle reference frames
- Post-processing explicit results





Advanced

ANSYS Linear & Non-Linear Dynamics



>> Duration

2 days

>> Participants

Mechanical Engineers,
Design Engineers

>>Prerequisites

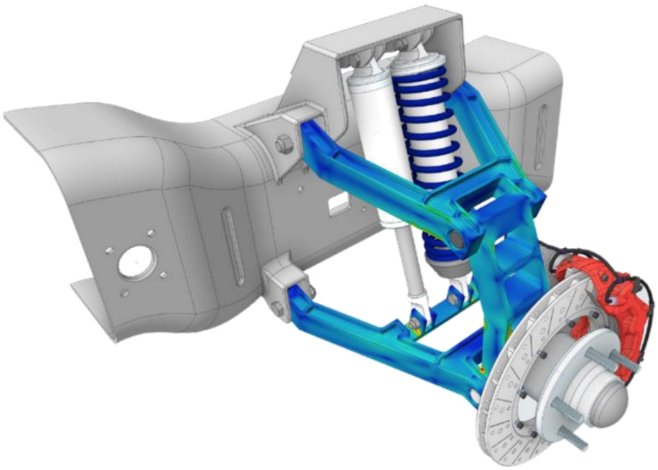
Engineering Knowledge
Fundamentals – FEA &
Dynamics
ANSYS Mechanical

Overview

This training course provide an overview of the various dynamic analysis types available with the ANSYS Mechanical and enable you to choose the type of analysis most pertinent to your needs, and understanding the results required for design decisions.

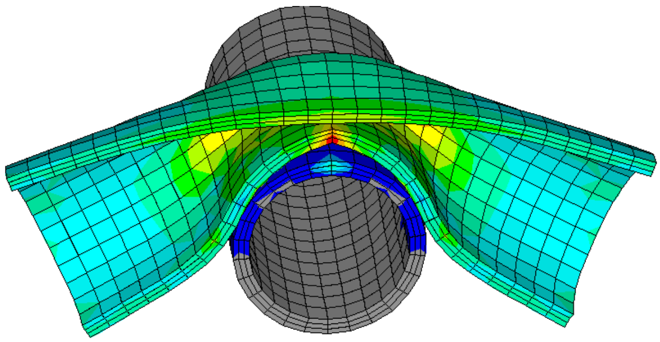
Topics

- Introduction to Dynamic analysis
- Modal analysis
- Linear perturbation
- Response spectrum Analysis
- Harmonic Analysis
- Random Vibration Analysis
- Transient Analysis



Advanced

ANSYS Structural Non-Linearities



>> Duration

2 days

>> Participants

Mechanical Engineers,
Design Engineers

>>Prerequisites

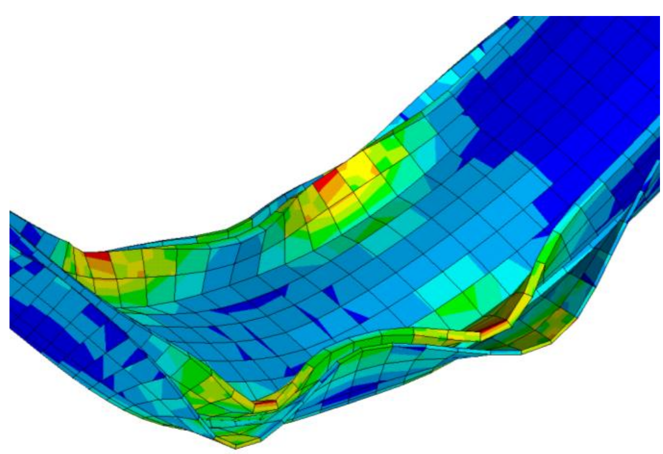
Engineering Knowledge
Fundamentals & Experience
with ANSYS Mechanical

Overview

This training course will give you an understanding of the nonlinear solution algorithm and procedures that must be used for nonlinear simulations, such as large deflection, surface-to-surface contact, elastic-plastic material models, nonlinear buckling, and others, while also helping you to identify the most typical reasons of convergence difficulties in non-linear solutions.

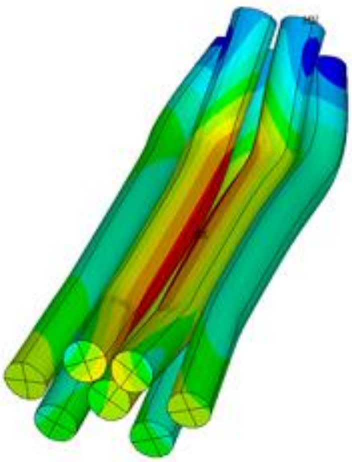
Topics

- 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



Advanced

ANSYS Non-linear Contacts



>> Duration

2 days

>> Participants

Mechanical Engineers,
Design Engineers

>>Prerequisites

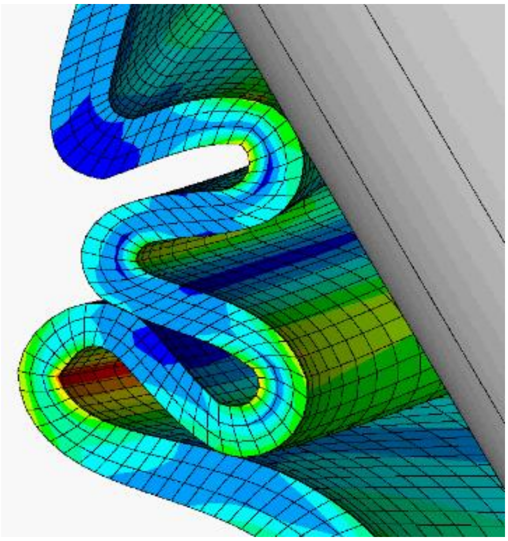
Engineering Knowledge
Fundamentals & Experience
with ANSYS Mechanical

Overview

The advanced connections between structural systems, including non-linear elements like seals, gaskets, interference fittings, bolt pretension, etc., will be covered in this training course. You'll also learn how to identify the most frequent causes of convergence issues in non-linear solutions.

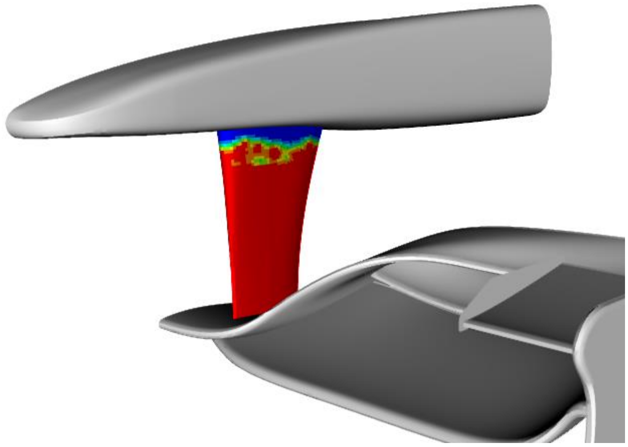
Topics

- Contact formulations
- Detection methods
- Trim contact
- Penetration Tolerance
- Contact Stiffness
- Pinball Region
- Symmetric and Asymmetric contact
- Body Types in Contact
- Postprocessing contact Results



Advanced

ANSYS Non-linear Materials



>> Duration

2 days

>> Participants

Mechanical Engineers,
Design Engineers

>>Prerequisites

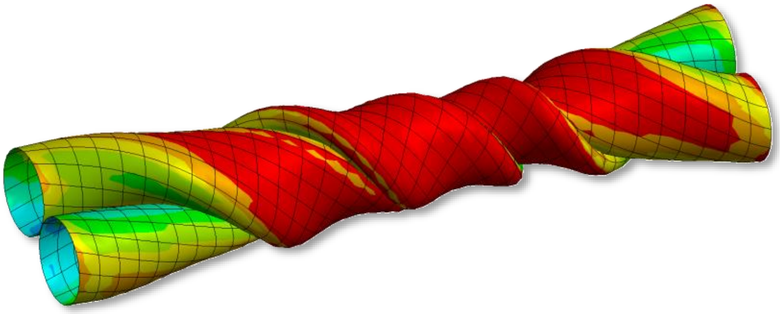
Engineering Knowledge Fundamentals & Experience with ANSYS Mechanical

Overview

The goal of this advanced course is to gain knowledge of and comprehension of the material models that can be used to represent non-linear material behavior in metals and elastomers.

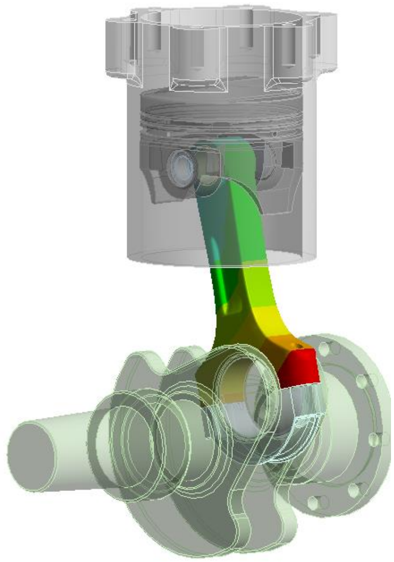
Topics

- Advanced Metal plasticity
- Creep
- Viscoplasticity
- Hyperelasticity
- Viscoelasticity
- Advanced Models



Advanced

ANSYS Rigid Body Dynamics



>> Duration

2 days

>> Participants

Mechanical Engineers,
Design Engineers

>>Prerequisites

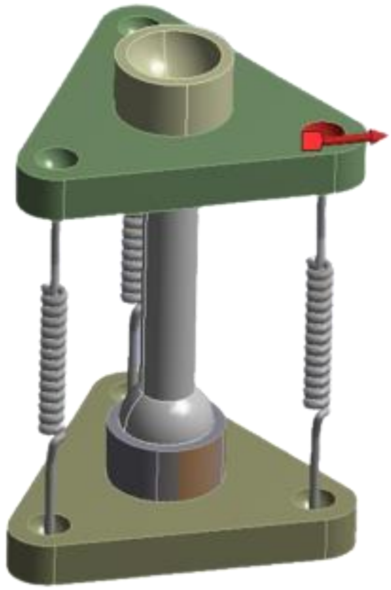
Engineering Knowledge
Fundamentals – FEA & Dynamics
ANSYS Mechanical

Overview

This training course demonstrates the power of rigid dynamics explicit solver for efficient and robust evaluation of mechanical systems containing complex assemblies of interconnected rigid parts undergoing large overall motion.

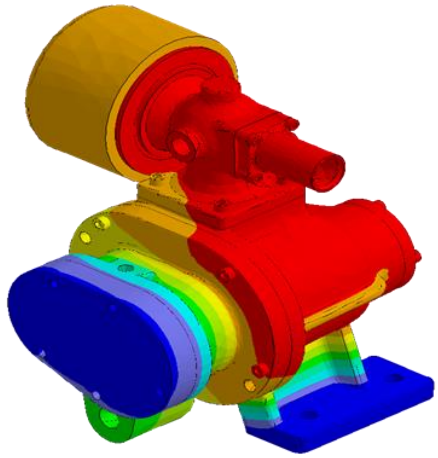
Topics

- Introduction To Rigid Body Motion
- Analysis Configuration Steps
- Connections
- Joints Definition
- Rigid/Flexible Multibody Dynamics
- Transient Structural
- Link With Control And System Simulation



Advanced

ANSYS Thermal Analysis



>> Duration

2 days

>> Participants

Mechanical Engineers,
Design Engineers

>>Prerequisites

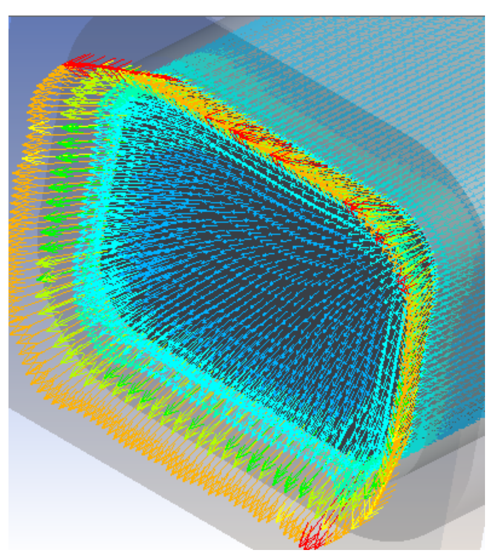
Engineering Knowledge Fundamentals & Experience with ANSYS Mechanical

Overview

The purpose of this course is to provide learners with the understanding and abilities needed to assess a structure's thermal response to various heat loads. This course covers methodologies for conducting complete solution processes for the three heat transfer phenomena under steady-state and time-varying conditions as well as thermal stress analysis.

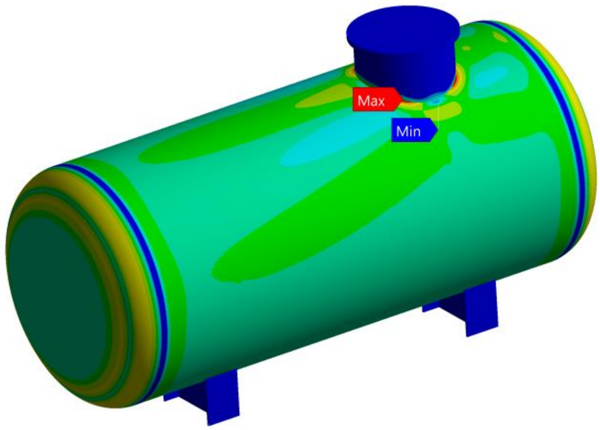
Topics

- Heat Transfer Fundamentals
- Preprocessing
- Boundary Conditions
- Steady State Heat Transfer
- Non-linear Thermal Analysis
- Transient Thermal Analysis
- Advanced Heat Transfer
- Thermal-structural Interaction



Advanced

ANSYS Fatigue Analysis Tool



>> Duration

2 days

>> Participants

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

>>Prerequisites

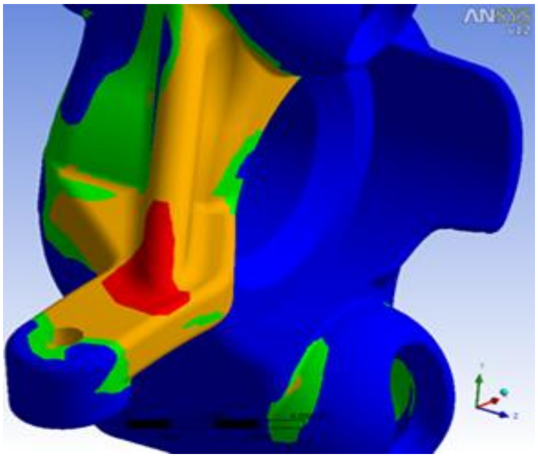
Engineering Knowledge Fundamentals, Experience with ANSYS Mechanical and theoretical background in fatigue analysis basics

Overview

This advanced training course helps learners how to select the best type of analysis for a given problem by understanding the stress-life and strain-life fatigue analysis methodologies for structures subject to fatigue loading. Also, the course will show the different types of fatigue loading on structures, including proportional, non-proportional, constant amplitude, and variable amplitude loadings.

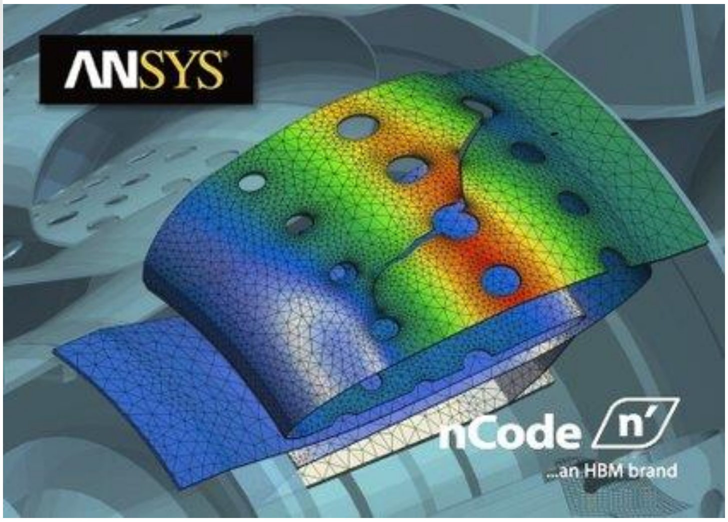
Topics

- Introduction To Fatigue Analysis
- High Cycle Fatigue
- SN Curves
- Low Cycle Fatigue
- Fatigue According To ASME Construction Code



Advanced

ANSYS nCode Design Life



>> Duration

2 days

>> Participants

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

>>Prerequisites

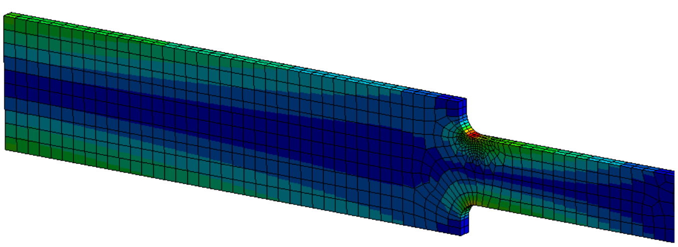
Engineering Knowledge Fundamentals, Experience with ANSYS Mechanical and theoretical background in fatigue analysis basics

Overview

This advanced training course helps learners Ansys nCode design life tool to perform stress-life and strain-life fatigue analysis methodologies for structures subject to fatigue loading. Also, the course will demonstrate Ansys nCode design life tool powerful material mapping. Load mapping and duty cycle techniques to create accurate and realistic models of complicated fatigue scenarios including thermal-structure analysis combination.

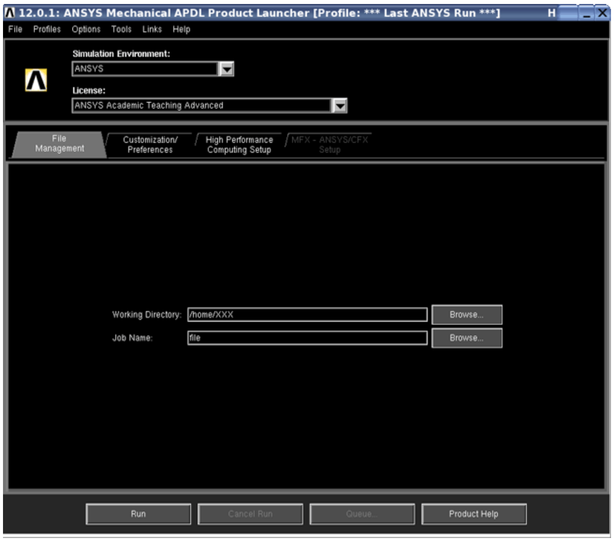
Topics

- Introduction To Fatigue Analysis
- SN Fatigue Analysis
- EN Fatigue Analysis
- Fatigue Analysis With Non-constant Loads
- Vibration Fatigue



Advanced

Introduction to ANSYS Mechanical APDL (Classic Environment)



>> Duration

4 days

>> Participants

Mechanical Engineers,
Design Engineers

>>Prerequisites

Engineering Knowledge Fundamentals however basic FEA knowledge will be useful

Overview

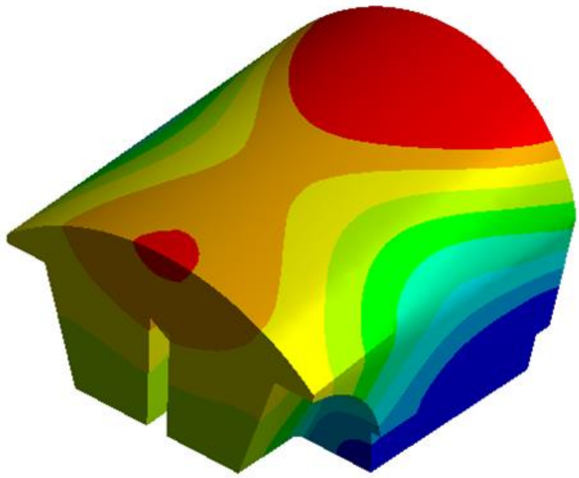
This training course covers an introduction to ANSYS Parametric Design Language (APDL) covering GUI and the associated workflow for using it to build, solve and post-process simulation models learning how to work at the node and element level of a finite element model, ensuring maximum control over its behavior

Topics

- Graphical User Interface
- Software General Parameters
- Introduction To FEA
- Import And Geometry Creation
- Elements Libarary
- Mesh Generation
- Materials Definition
- Loads Definition
- Getting A Solution And Solver Selection
- Static, Thermal And Modal Analysis (Setup, Solving And Postprocessing)
- Parameters
- Constrain Equations, Coupling And Contact Creation

Advanced

Acoustics Analysis in ANSYS Mechanical



>> Duration

2 days

>> Participants

Mechanical Engineers,
Design Engineers

>>Prerequisites

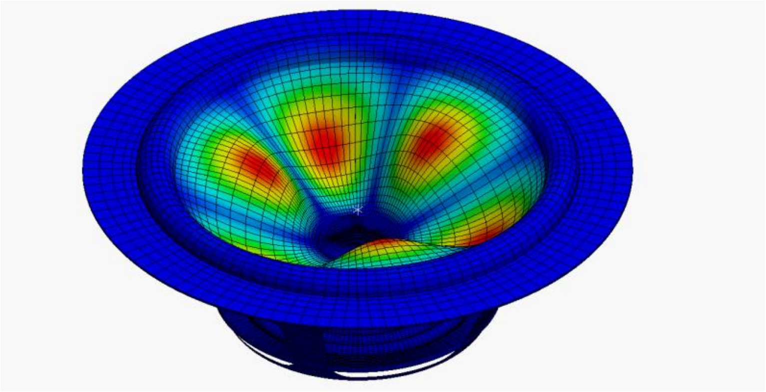
Engineering Knowledge
Fundamentals & Experience
with ANSYS Mechanical

Overview

This training course will provide insight on how to use Ansys Mechanical to perform Acoustics analysis. It will provide a general understanding of acoustic phenomenon, terminology, and governing equations, while introducing 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



Advanced

Introduction Ansys ACT

>> Duration

2 days

>> Participants

Mechanical Engineers, Design Engineers

>>Prerequisites

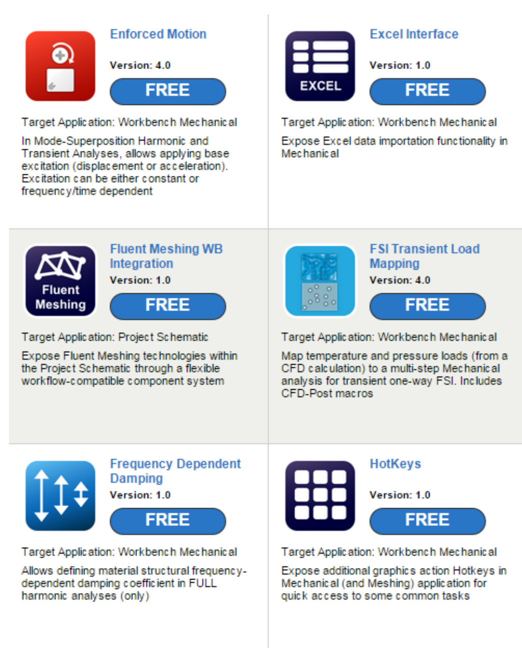
Engineering Knowledge Fundamentals, Experience with ANSYS Mechanical & basic knowledge of Python programming is recommended

Overview

This training course is to cover Ansys Application Customization Toolkit (ACT) in Mechanical Workbench to Learn to automate the creation of standard tree objects in Mechanical or its integrated modules, create custom loads and results and understand the capabilities of ACT.

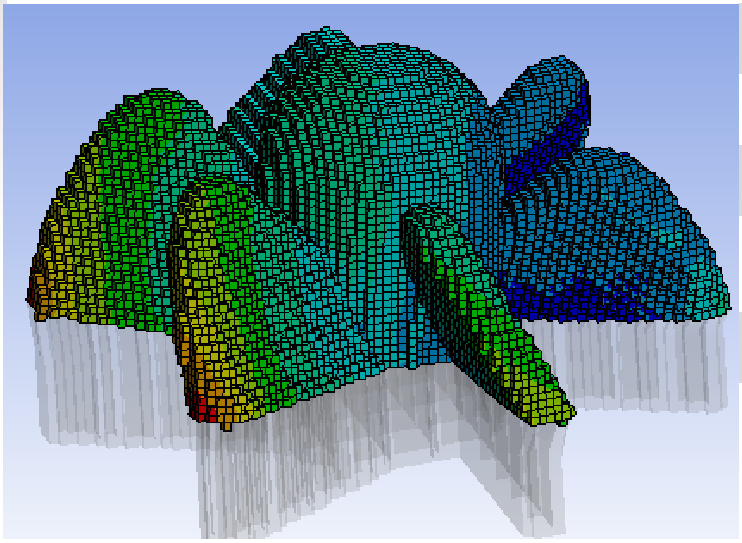
Topics

- Introduction To ACT
- Applications Examples
- XML And Python Programming
- IronPython Console
- Macros Integration
- Development Of Customized Results
- Advanced Functionalities
- Practical Cases



Advanced

Introduction to ANSYS Additives



>> Duration

2 days

>> Participants

Mechanical Engineers,
Design Engineers

>>Prerequisites

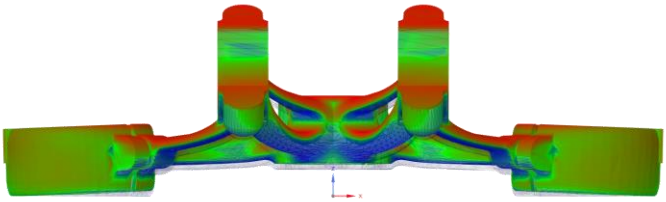
Engineering Knowledge
Fundamentals – FEA
ANSYS Mechanical
Ansys Spaceclaim

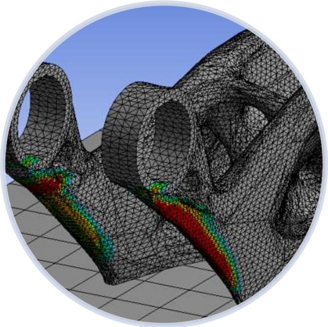
Overview

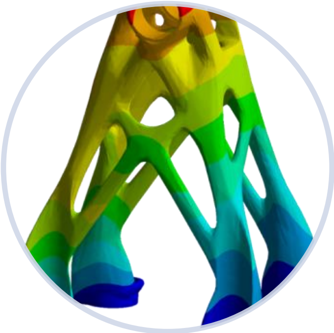
This is an introductory to intermediate level training to provide knowledge about the 3D print process simulation. It will give you a general overview of the challenges when simulating a print process.

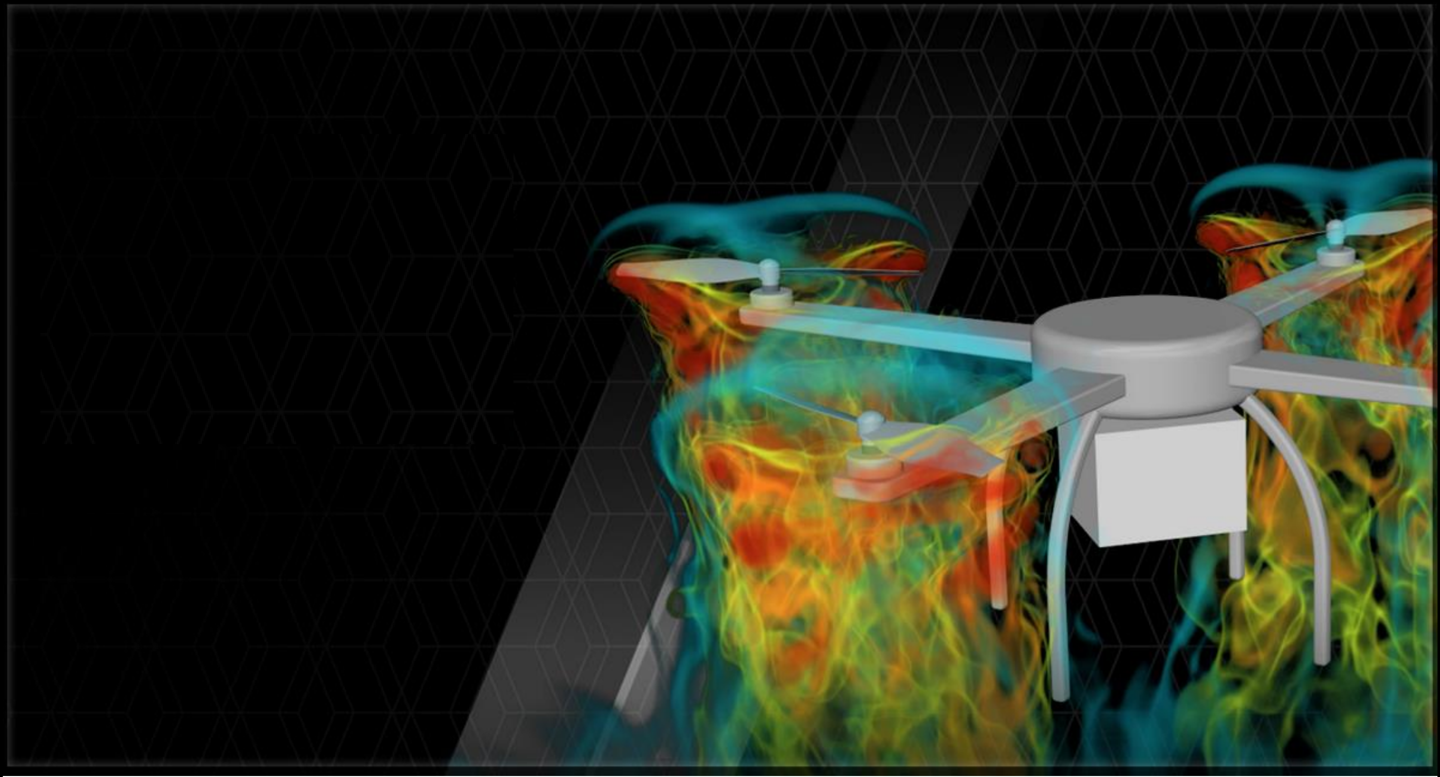
Topics

- 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

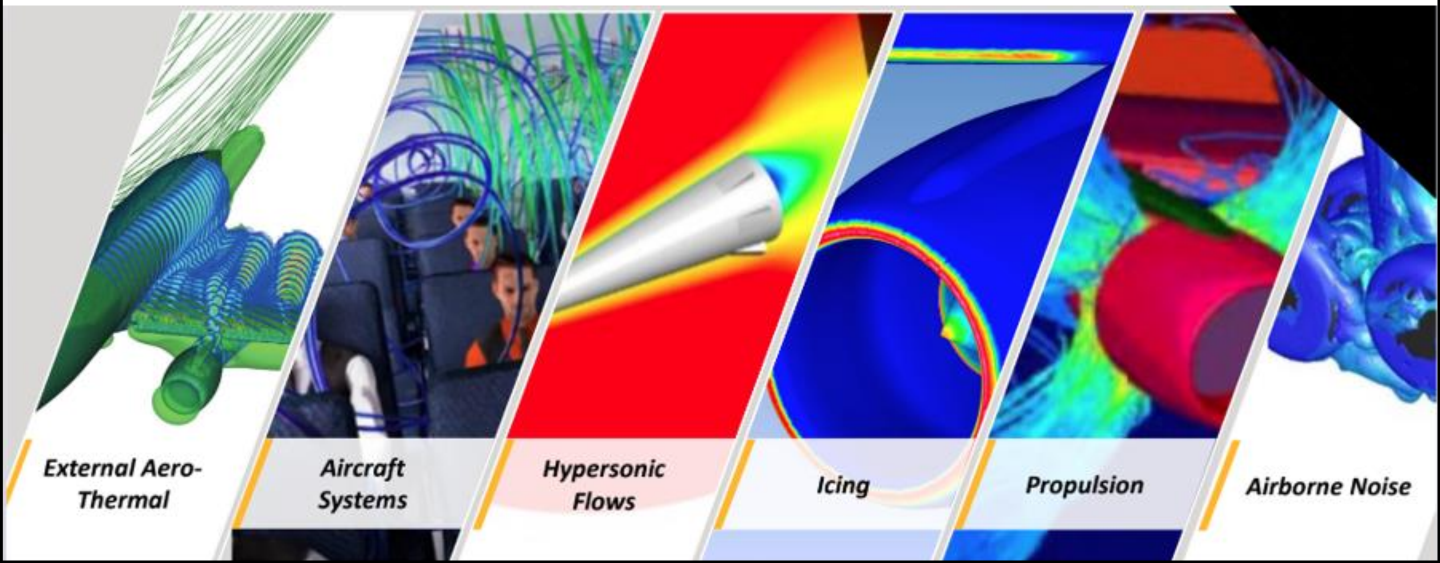






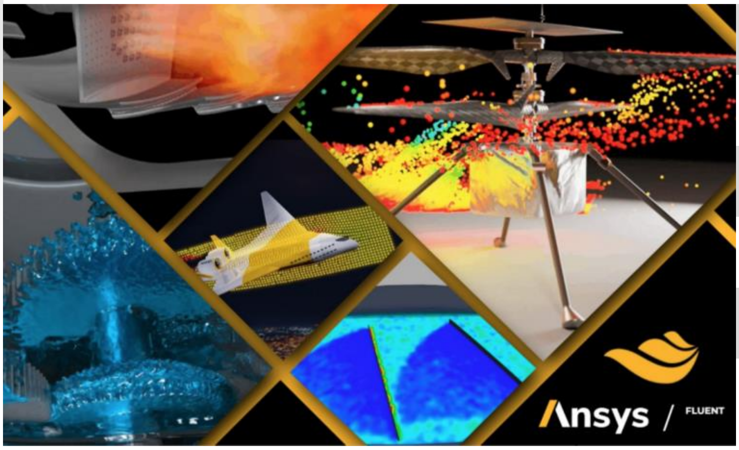


Fluids


External Aero-Thermal
Aircraft Systems
Hypersonic Flows
Icing
Propulsion
Airborne Noise

Standard

Introduction to Ansys Fluent



>> Duration

4 days

>> Participants

Engineers and Designers

>>Prerequisites

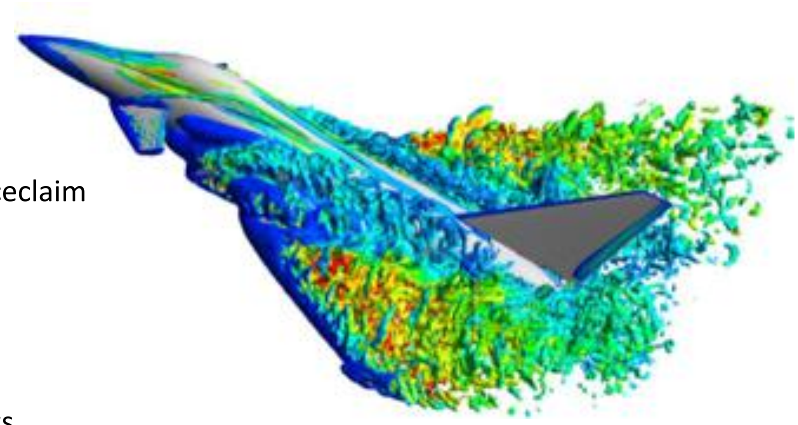
Technical education and/or background in fluids

Overview

The aim of this training program is to equip participants with the fundamental knowledge and skills necessary for Computational Fluid Dynamics (CFD) modeling using Ansys Fluent solutions. The course covers the entire process of conducting a CFD simulation, from geometry preparation and mesh generation, to defining the problem, solving it, and post-processing the results. Additionally, the training program offers insights into best practices for conducting popular CFD simulations, as well as advanced post-processing features. By the end of the program, participants will have a solid understanding of CFD simulations, and be able to apply this knowledge to solve fluid dynamics problems in the real world.

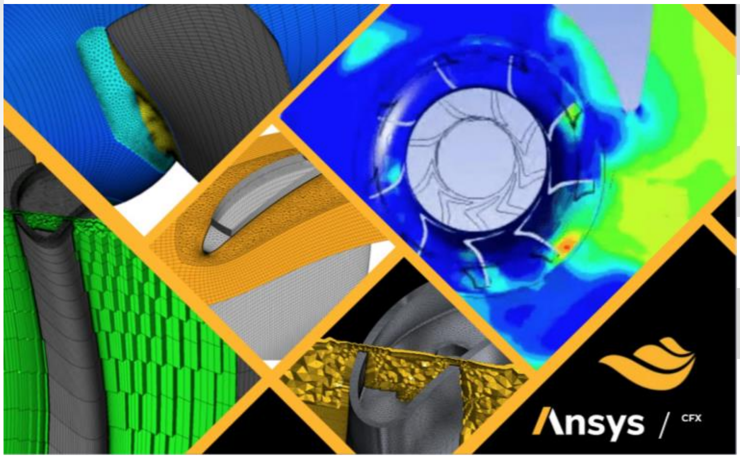
Topics

- Overview of CFD Process
- Geometry preparation using Spaceclaim
- Meshing using Fluent Meshing
- Solving using Ansys Fluent
- Results and postprocessing
- Parametrization and Design Points



Standard

Introduction to Ansys CFX



>> Duration

4 days

>> Participants

Engineers and Designers

>>Prerequisites

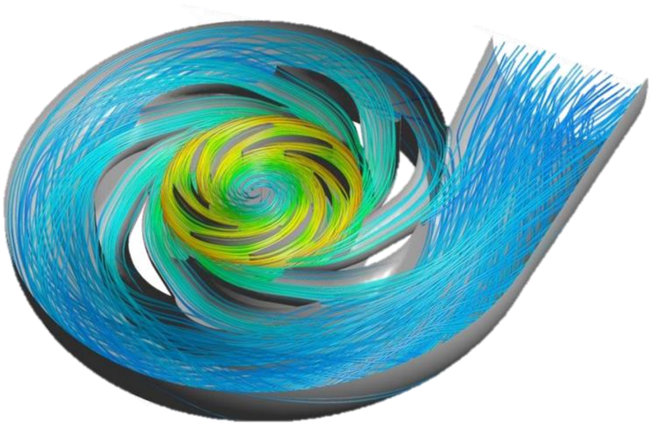
Technical education and/or background in fluids

Overview

This training program provides participants with the essential skills required to perform basic computational fluid dynamics (CFD) simulations using Ansys CFX software. The program covers the complete CFD workflow, including working with CAD models in Ansys Spaceclaim, creating high-quality CFD meshes with Ansys Fluent Meshing, and 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 perform basic CFD simulations using Ansys CFX software.

Topics

- Overview of CFD Process
- Geometry preparation using Spaceclaim
- Meshing using Ansys Meshing
- Solving using Ansys CFX
- Results and postprocessing
- Parametrization and Design Points



Standard

CHEMKIN-Pro & Model Fuel Library

>> Duration

4 days

>> Participants

Engineers and Designers

>>Prerequisites

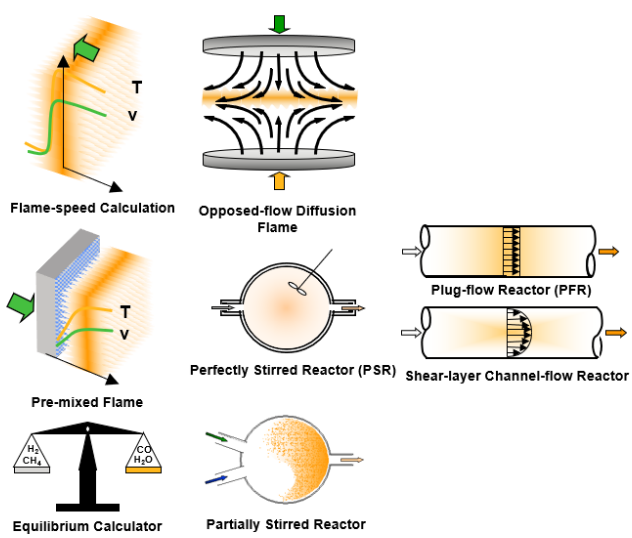
Technical education and/or background in fluids

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.

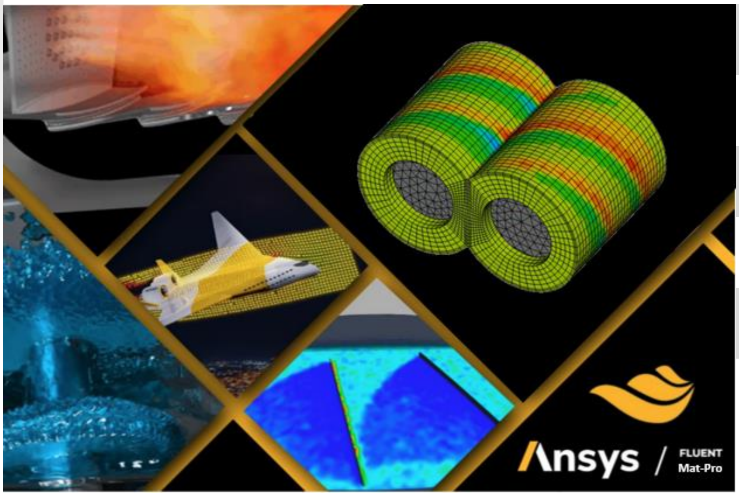
Topics

- Introduction and Chemistry Fundamentals
- Combustion and Emission Modeling
- Catalysis and Materials
- Multiphase Processes Framework



Standard

POLYFLOW is now Mat-Pro



>> Duration

2 days

>> Participants

Engineers and Designers

>>Prerequisites

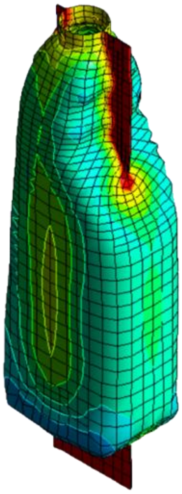
- Technical background-Fluids
- Pre-processing-Spaceclaim
- Ansys/Fluent Meshing

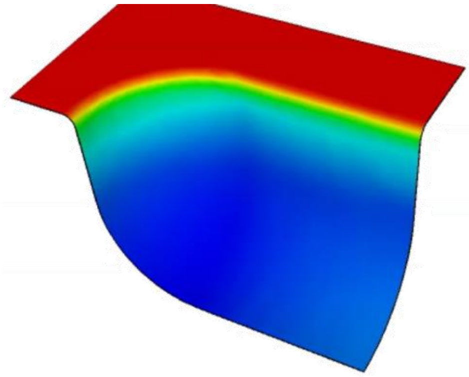
Overview

Ansys Polyflow is a powerful CFD software used in manufacturing applications. This training course will teach participants how to effectively use Ansys Polyflow to simulate complex flows, identify potential manufacturing issues, and develop solutions to optimize product quality and reduce production costs. By completing this training, companies can improve their work efficiency and enhance their team's skills, leading to better manufacturing practices and a stronger bottom line. The Fluent Materials Processing Workspace will also be covered, providing participants with a comprehensive understanding of how to utilize the latest manufacturing tools to solve real-world problems.

Topics

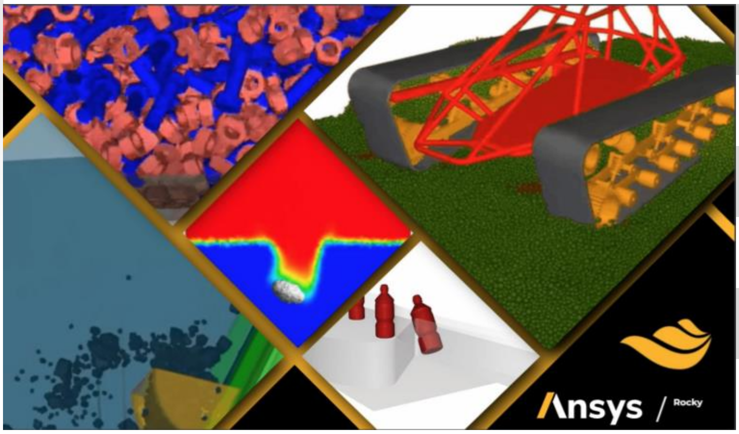
- Ansys Polyflow Overview
- 2.5D Extrusion
- 3D Extrusion
- Modeling Forming
- Time dependent flows
- Workbench parametrization
- Post processing using CFD-Post





Standard

Rocky



>> Duration

3 Days

>> Participants

Engineers and Designers

>>Prerequisites

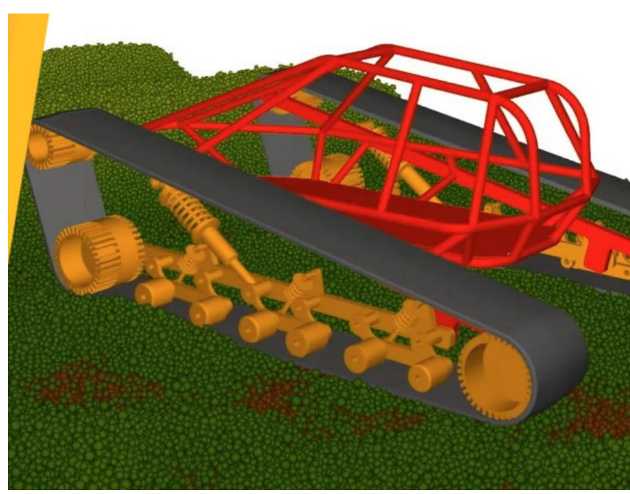
Engineering Knowledge -Fluids

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.

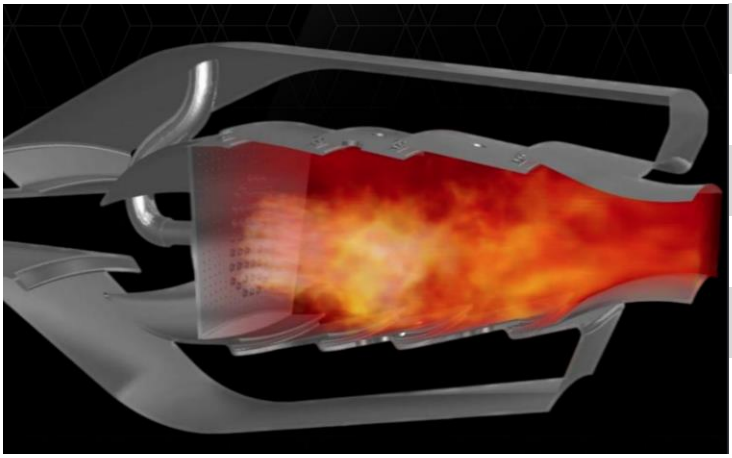
Topics

- Introduction to DEM and Basic Concepts
- Particle Shapes and Motion Frames
- Post-processing in Rocky DEM
- Additional Contact Force Models and Coarse Grain Model



Advanced

Ansys Fluids – Advanced



>> Duration

Customized

>> Participants

Engineers and Designers

>>Prerequisites

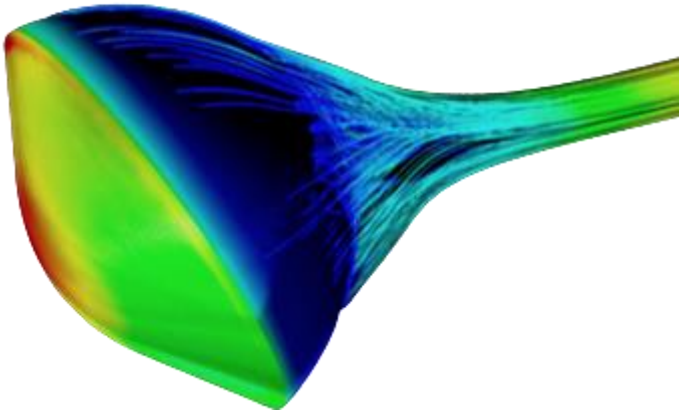
Engineering Knowledge -Fluids
Ansys Fluent/Ansys CFX

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 to reflect in their deliverables. 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.

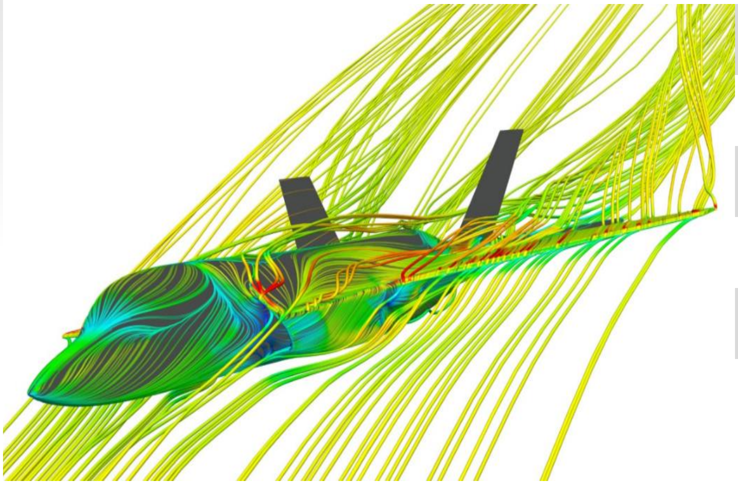
Topics

- Aeroacoustics Modeling
- Combustion Modeling
- Multiphase Modeling
- Turbulence Modeling
- Turbomachinery Modeling
- Fluid Structure Interaction FSI Modeling
- Advanced DPM - Spray Modeling



Advanced

External Aerodynamics



>> Duration

1 day

>> Participants

Engineers and Designers

>>Prerequisites

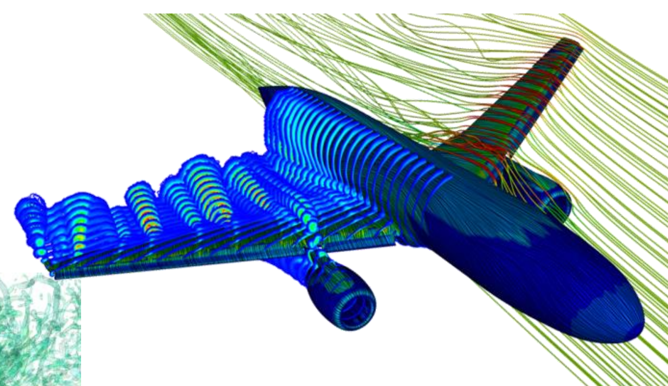
Engineering Knowledge -Fluids
Ansys Fluent
Aerodynamics

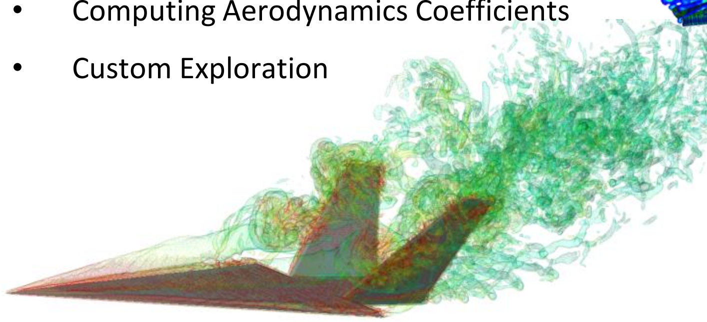
Overview

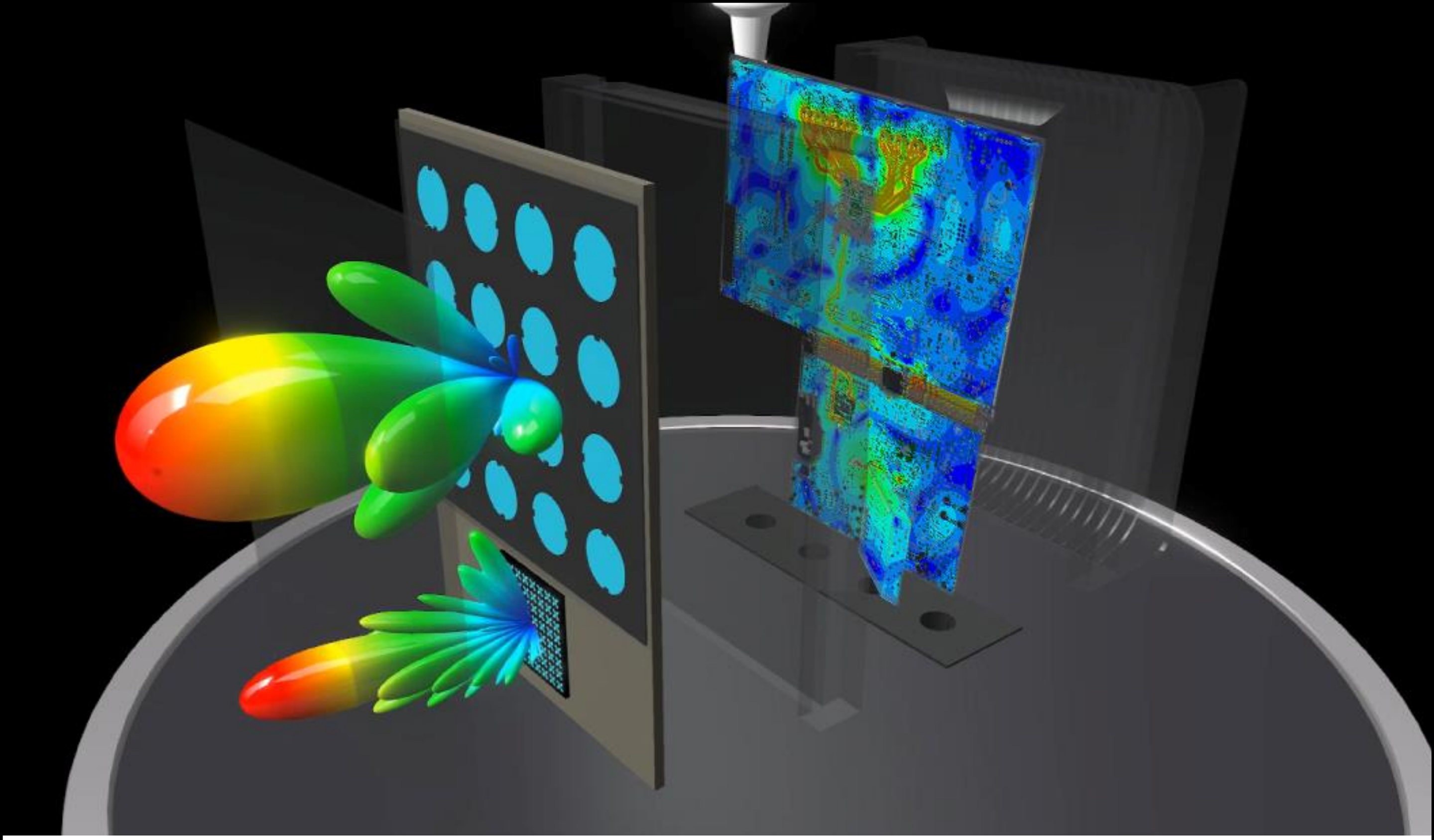
Ansys Fluent Aero Workspace is a powerful tool for aerospace and defense engineers, providing customized workflows to access automation enhancements, High-Speed Numeric (HSN), and convergence best practices. This training is designed to cover all aspects of the Fluent Aero Workspace, from creating a 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

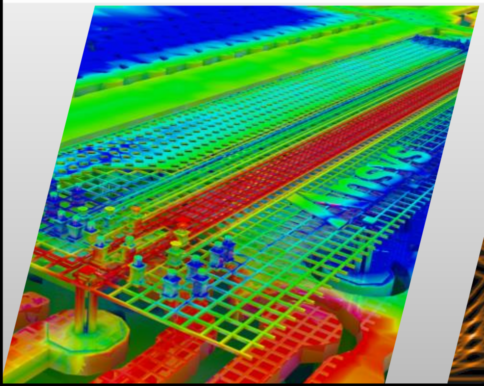
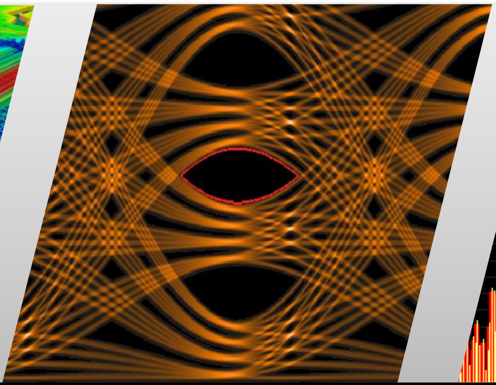
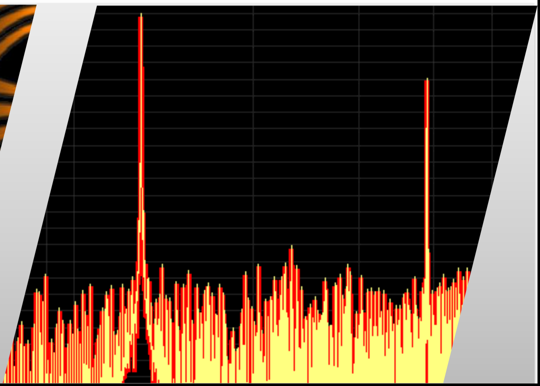
- Fluent Aero Overview
- Parametric Setup
- Viewing the Results inside Fluent Aero
- Computing Aerodynamics Coefficients
- Custom Exploration





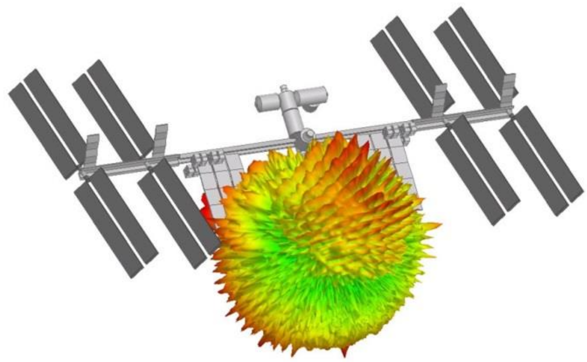


Electromagnetics

Standard

Introduction to Ansys HFSS



>> Duration

2 days

>> Participants

Engineers and Designers (HF)

>>Prerequisites

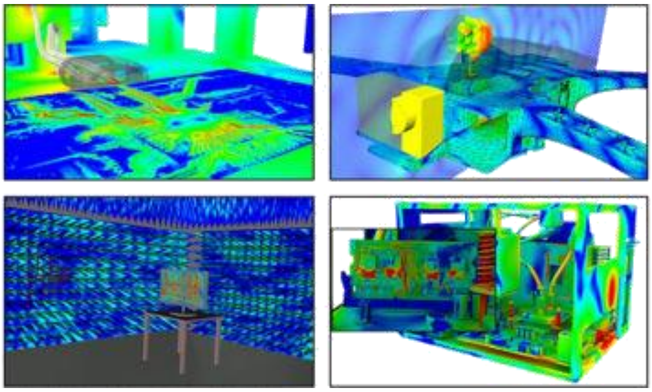
Technical education and/or background in high frequency electromagnetics

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.

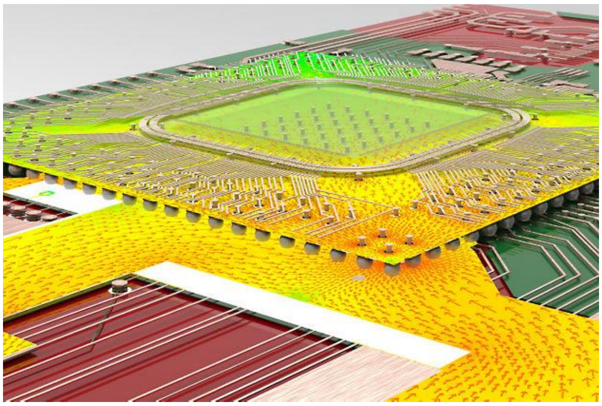
Topics

- 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
- Wrap up



Standard

Introduction to Ansys SIwave



>> Duration

2 days

>> Participants

Electronics engineers involved in high speed PCB design

>>Prerequisites

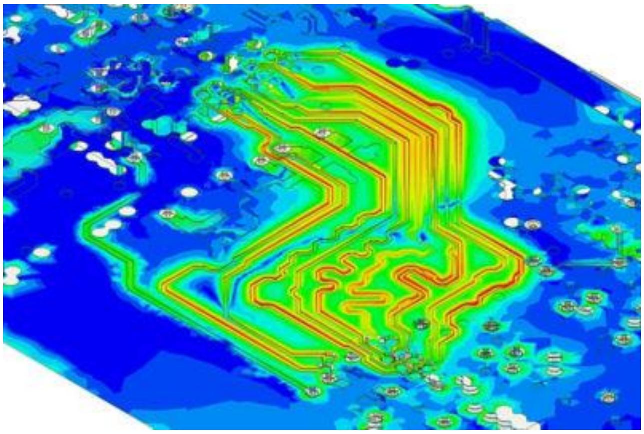
Familiarity with PCB layouts
Familiarity with high-speed digital signal electronic engineering

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.

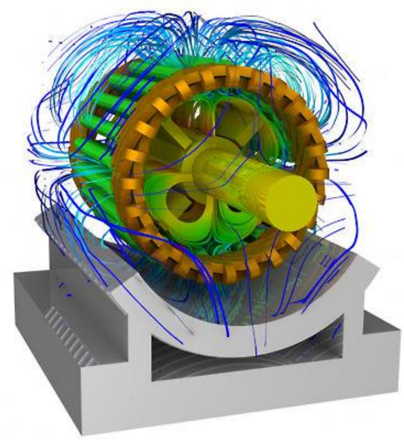
Topics

- SIwave overview
- Import and scans
- Signal integrity
- Power integrity
- Wrap up



Standard

Introduction to Ansys Maxwell



>> Duration

2 days

>> Participants

Engineers and Designers (LF)

>>Prerequisites

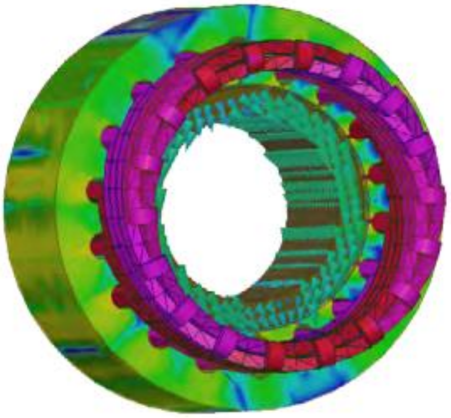
Knowledge of static and quasistatic EM

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.

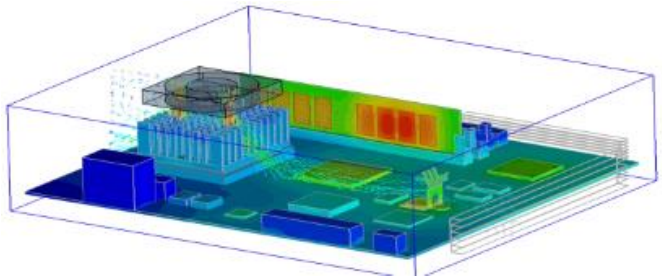
Topics

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



Standard

Introduction to Ansys Icepak



>> Duration

2 days

>> Participants

Thermal, Mechanical and Electrical engineers

>>Prerequisites

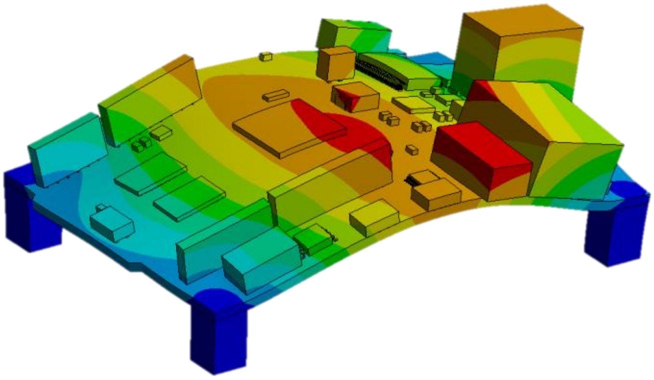
Technical education in electronics and/or fluid mechanics and heat transfer

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.

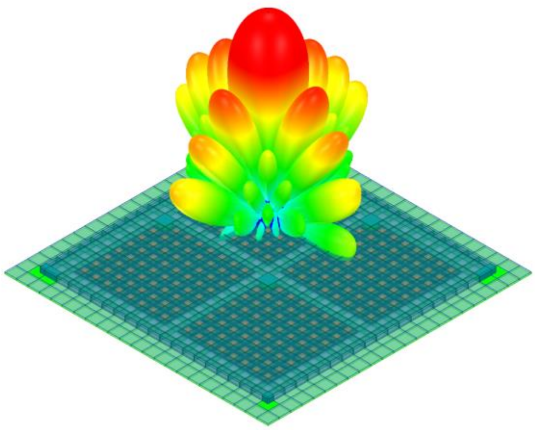
Topics

- Icepak in AEDT Overview
- Interface and modeling basics
- Meshing
- Solution setup and post-processing
- Electro-thermal analysts
- Wrap up



Advanced

Antenna Array and Periodic Structures



>> Duration

1 days

>> Participants

Engineers and Designers

>>Prerequisites

Technical education and/or background in high frequency electromagnetics

Overview

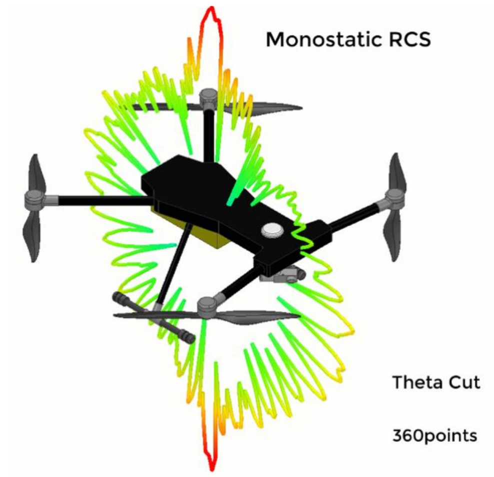
The HFSS Antenna Arrays and Periodic Structures course explores topics such as Infinite Array (Unit Cell analysis), Finite Array with both periodic and semi-periodic (non-identical) unit cell structures, Scanning the beams using Finite Array Beam Angle Calculator Toolkit, and Frequency Selective Surfaces (FSS). This course is designed for intermediate-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

Advanced

Ansys HFSS SBR+ Radar Cross Section (RCS)



>> Duration

1 days

>> Participants

Engineers and Designers

>>Prerequisites

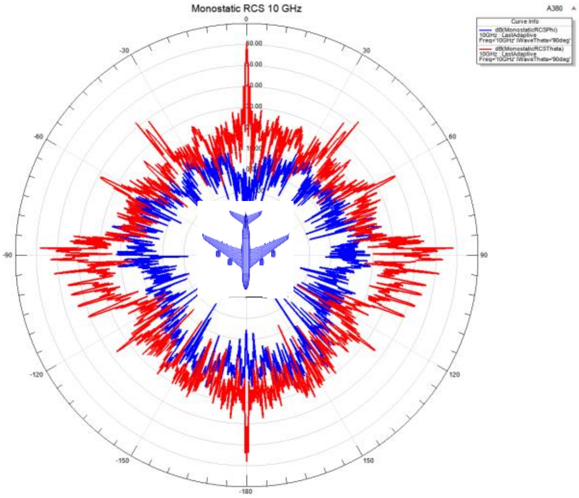
Technical education and/or background in high frequency electromagnetics

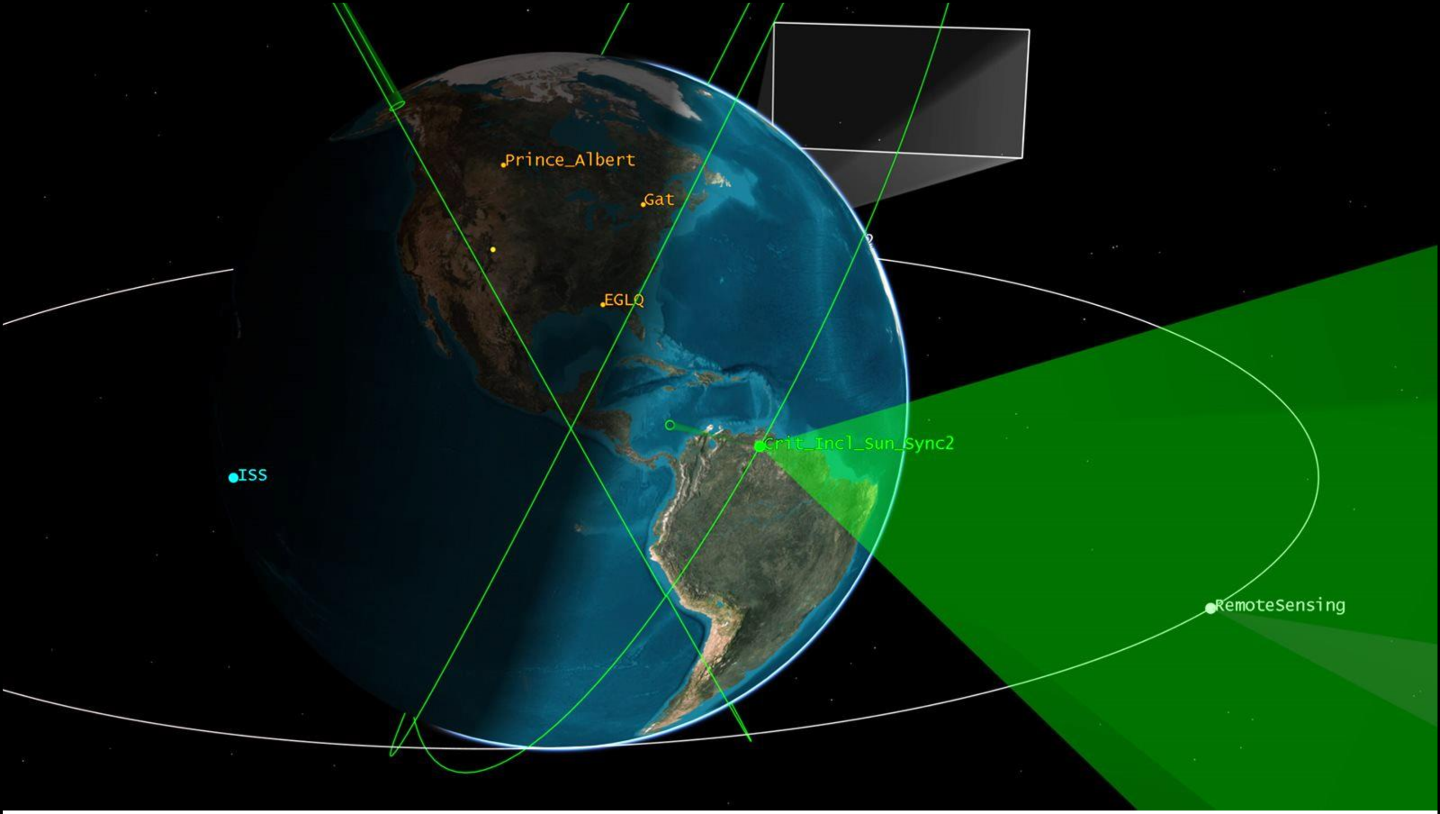
Overview

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..

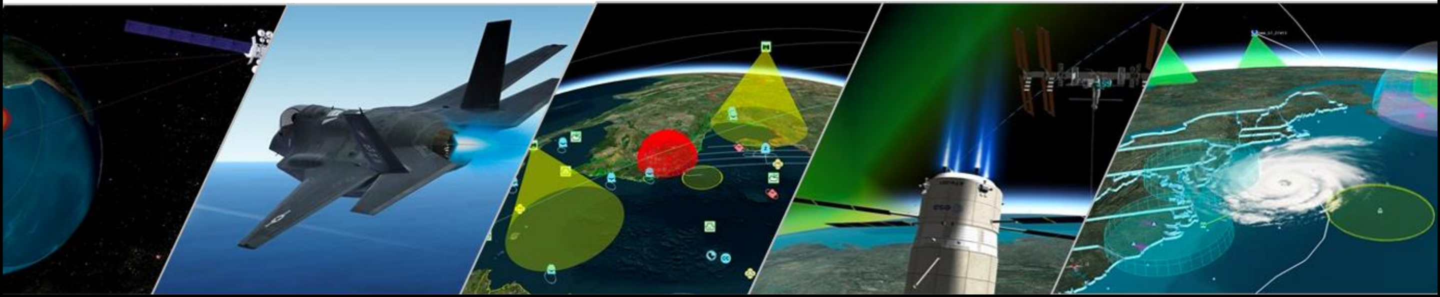
Topics

- 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



Standard

Introduction to Ansys STK

>> Duration

3 days

>> Participants

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

>>Prerequisites

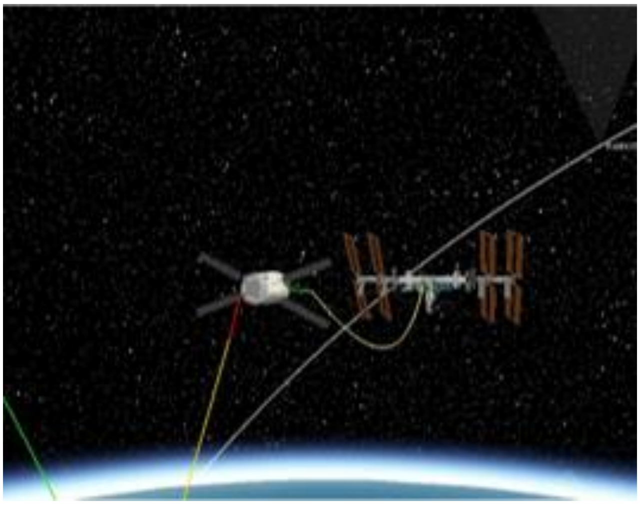
Technical education and/or background in aerospace.

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.

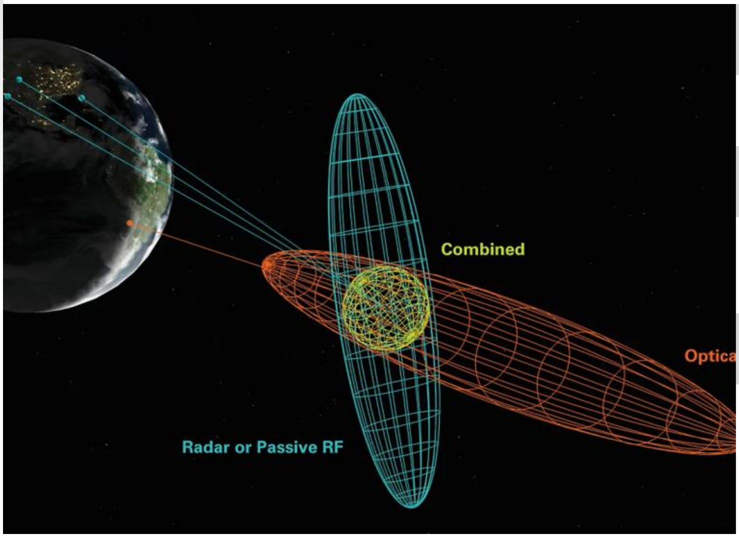
Topics

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



Standard

Introduction to Ansys ODTK



>> Duration

3 days

>> Participants

Aerospace Engineers, Satellite Operators, System Engineers.

>>Prerequisites

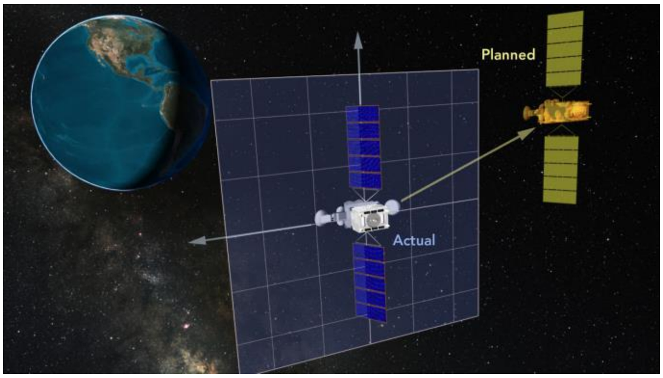
Technical education and/or background in aerospace and orbit determination.

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

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 ANSYS product, consider Fluid Codes customized training. This solution is more tailored to your own needs, it enables you to spend 1:1 time with an experienced engineer looking in detail at your own application and its specific challenges, and it helps you with the usage of your ANSYS product.

Topics

- Customized

Contact us

training@fluidcodes.com

www.fluidcodes.com

FLUID CODES

Ansys / ELITE CHANNEL PARTNER

UAE
Office 901, Jumeirah
Bay X2, Cluster X,
Jumeirah Lake Towers
Dubai, UAE
Tel: +971 (0) 4 3308666

Saudi Arabia
Office number 126
Regus Novotel
Business Park
Dammam 31413, KSA
Tel: +966 13 845 2603

Bulgaria
Office 807, 8th Floor
205 Alexander
Stamboliyski blvd
Sofia, Bulgaria
Tel: +359 877 43 59 58

Egypt
Sara's Tower-23, 3rd
Floor, Batrawy street
Nasr city
Cairo, Egypt
Tel: +2 2 2617124

www.fluidcodes.com