MSC **Training** Catalog

Comprehensive Training For Your Learning Needs

PART OF HEXAGON

MSC Software

Table of Contents

About MSC Software 5
MSC Products 6
Instructor-Led Training 7
E-Learning Subscription 8

Actran 9

ACT140 - Launch Acoustics with Actran 9

Adams 10

ADM701 - Complete Multibody Dynamics Analysis with Adams 10
ADM702 - Fundamentals of Multibody Dynamics Analysis with Adams 10
ADM703A - Advanced Modeling Elements and Techniques with Adams/Solver 11
ADM703B - Adams/Solver Theory: Achieving Robust, Converged Solutions 11
ADM703C - Writing User Subroutines in Adams/Solver 12
ADM704A - Advanced Parametrics, Design Sensitivity, and Optimization using Adams/View 12
ADM704B - Automating Tasks using Adams/View Scripting, Macros, and GUI Customization 13
ADM710 - Flex Body Dynamics and Modal Stress Recovery using Adams 13
ADM711 - Control System Integration with Adams using MATLAB or Easy5 14
ADM720 - Frequency Domain Analysis using Adams/Vibration 14
ADM730 - Design of Experiments (DOE) and Stochastics (Monte Carlo) Analysis using Adams 15
ADM740 - Vehicle Modeling and Simulation using Adams/Car 15
ADM741 - Vehicle Modeling and Simulation using Adams/Driveline 16
ADM750 - Gear, Belt, and Chain Modeling with Adams/Machinery 16
ADM761 - Basic Suspension and Full Vehicle Analysis using Adams/Chassis 17

Digimat 18

DIG101 - Digimat Introductory Training - Structural Engineering 18
DIG102 - Digimat Introductory Training - Material Engineering 18
DIG103 - Digimat Introductory Training - Material & Structural Engineering 18
DIG120 - Digimat Standard Training - Material & Structural Engineering 18
DIG201 - Digimat Standard Training - Structural Engineering 19
DIG202 - Digimat Standard Training – Material Engineering 19
DIG301 - Digimat Advanced Training - Aerospace Application 19
DIG325 - Digimat Advanced Training - Electrical/Electronic Application 20
DIG340 - Digimat Advanced Training - Automotive Application 20

Dytran 21

DYT101 - Introduction to Dytran 21
DYT103 - Introduction to Airbag Analysis and Occupant Safety using Dytran 22

Easy5 23

EAS101 - Dynamic System Modeling and Simulation using Easy5 23
EAS103 - Modeling and Simulation of Fluid Power Systems using Easy5 23
EAS105 - Modeling and Simulation of Gas Systems using Easy5 24
EAS107 - Modeling and Simulation of Multi-Phase Fluids using Easy5 24

Table of Contents

Flightloads 25

FLD120 - Aeroelasticity using FlightLoads and Patran 25
FLD120S - FlightLoads and Aeroelasticity - Static Analysis 25

Marc 27

MAR101 - Basic Nonlinear Analysis using Marc and Mentat 27
MAR102 - Advanced Nonlinear Analysis using Marc and Mentat 27
MAR103 - Elastomer Analysis using Marc and Mentat 27
MAR106 - Electromagnetic Analysis using Marc and Mentat 28
MAR120 - Basic Nonlinear Analysis using Marc and Patran 28
MAR121 - Advanced Nonlinear Analysis using Marc and Patran 29

MaterialCenter 30

MAT101 - Introduction to MaterialCenter 30
MAT102 - MaterialCenter for the Materials Engineer 30

MSC Nastran 31

NAS101A - Linear Static and Normal Modes Analysis using MSC Nastran 31
NAS101B - Advanced Linear Analysis using MSC Nastran 31
NAS102A - Dynamic Analysis using MSC Nastran 31
NAS102B - Advanced Dynamic Analysis using MSC Nastran 32
NAS103 - MSC Nastran Nonlinear Analysis 32
NAS104 - Thermal Analysis using MSC Nastran (SOLs 153 and 159) 33
NAS106A - Basic Substructure Analysis using MSC Nastran - Primary Superelements 34
NAS106B - Advanced Substructure Analysis using MSC Nastran - Secondary Superelements 34
NAS107 - Design Sensitivity and Optimization using MSC Nastran 34
NAS110 - Working with Custom MSC Nastran Solution Sequences using DMAP 35
NAS111 - Aeroelasticity using MSC Nastran 36
NAS113 - Composite Material Analysis with MSC Nastran 37
NAS115 - Fluid Structure Analysis using MSC Nastran 37
NAS120 - Linear Static Analysis using MSC Nastran and Patran 38
NAS122 - Basic Dynamic Analysis using MSC Nastran and Patran 38
NAS123 - Implicit Nonlinear Analysis using MSC Nastran (SOL600) 39
NAS124 - Thermal Analysis using MSC Nastran (SOL400) 39
NAS126 - Explicit Nonlinear Analysis (SOL700) using MSC Nastran and Patran 40
NAS127 - Rotordynamic Analysis using MSC Nastran 40
NAS133 - Contact Analysis using MSC Nastran and Patran 41
NAS134 - Advanced Contact Analysis using Contact Pairs in MSC Nastran and Patran 41
NAS318 - Implementation of Fatigue Methods using MSC Nastran Embedded Fatigue with Patran 41
NAS319A - Fatigue & CAE Intergration 42
NAS319B - Fatigue & Dynamics – The Reliability of Vibrating Systems 42
NAS319C - Practical Implementation of Fatigue Methods with MSC Nastran, Patran and Other Applications 43
NAS400 - Implicit Nonlinear Analysis using MSC Nastran and Patran 43

Table of Contents

MVision 44

MVI320 - MVision Introduction 44

Patran 45

PAT301 - Introduction to Patran 45
PAT302 - Advanced Geometry, Meshing, Customization, and Variable LBCs using Patran 45
PAT304 - Automating Tasks and Basic GUI Customization
Using the Patran Programming Command Language (PCL) 46
PAT312 - Thermal Analysis using Patran Thermal 46
PAT318A - Basic Durability and Fatigue Life Analysis using MSC Fatigue 47
PAT318B - Advanced Durability and Fatigue Life Analysis using MSC Fatigue 47
PAT325 - Composite Laminate Modeling using Patran 47

SimDesigner 49

SMD102 - Fundamentals of Multibody Dynamics Analysis with SimDesigner 49

SimManager 50

SMM101 - Introduction to SimManager 50
SMM102 - SimManager Basic Configuration 50
SMM111 - Introduction to the SimManager Automotive Solution 51
SMM112 - SimManager Automotive Solution Configuration 51
SMM201 - SimManager Installation and Administration 52

Simufact 53

SMF101 - Introduction to Simufact Forming 53
SMF102 - Simulation of mechanical joining processes using Simufact Forming 53
SMF201 - Introduction to Simufact Welding 54

SimXpert 55

SMX101 - Introduction to SimXpert FE Workspaces 55
SMX113 - Composite Material Analysis using SimXpert 55
SMX120 - Linear Statics, Normal Modes, and Buckling Analysis using SimXpert 56
SMX121 - Complete Multibody Dynamics Analysis with SimXpert Motion 56
SMX122 - Dynamic Analysis using SimXpert 57
SMX124 - Thermal Analysis (SOL400) using SimXpert 57
SMX126 - Explicit Nonlinear Analysis using SimXpert 58
SMX131 - SimXpert Templates 58
SMX132 - SimXpert Scripting 59
SMX140 – Dynamic System Modeling and Simulation using SimXpert 59
SMX400 – Nonlinear Structural Analysis using SimXpert 60
SMX702 – Fundamentals of Multibody Dynamics Analysis with SimXpert 60

Sinda 61

SND501 - Network Thermal Analysis using MSC Sinda 61
SND502 - Thermal Analysis using Patran with MSC Sinda 61
SND503 - Spacecraft Thermal Analysis using THERMICA V4 62

Contacts 63

About MSC Software

Simulating Reality, Delivering Certainty

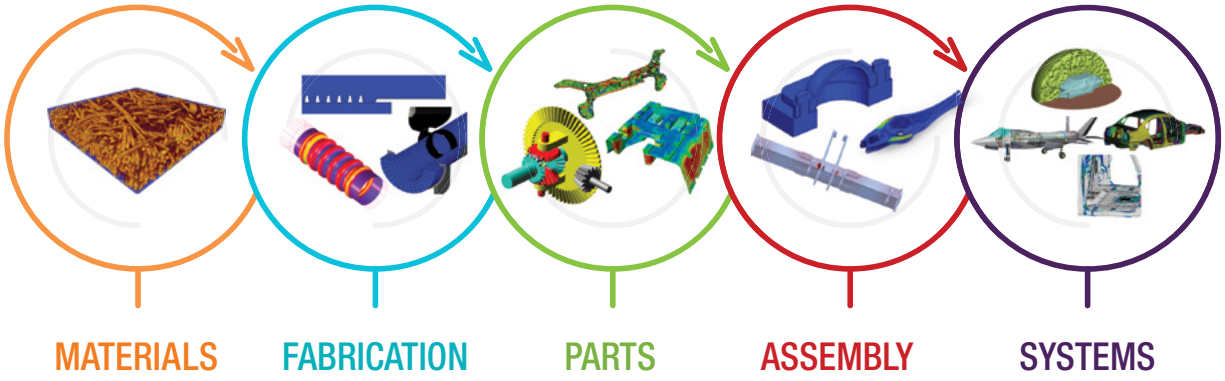
MSC Software is one of the ten original software companies and the worldwide leader in multidiscipline simulation. As a trusted partner, MSC Software helps companies improve quality, save time and reduce costs associated with design and test of manufactured products. Academic institutions, researchers, and students employ MSC technology to expand individual knowledge as well as expand the horizon of simulation. MSC Software's engineering simulation technology is used by leading manufacturers for linear and nonlinear finite element analysis (FEA), acoustics, fluid-structure interaction (FSI), multi-physics, optimization, fatigue and durability, multi-body dynamics, and control systems simulation. The company's products accurately and reliably predict how products will behave in the real world to help engineers design more innovative products - quickly and cost effectively.

MSC Software Corporation was formed in 1963 and was awarded the original contract from NASA to commercialize the finite element analysis (FEA) software known as Nastran (NASA STRuctural ANalysis). MSC pioneered many of the technologies that are now relied upon by industry to analyse and predict stress and strain, vibration & dynamics, acoustics, and thermal analysis in our flagship product, MSC Nastran. Today MSC is a privately owned company of the Symphony Technology Group and provides an extensive range of CAE solutions.

Solutions

Over our rich history, MSC has developed or acquired many other well-known CAE applications including Patran, Adams, Marc, Dytran, Fatigue, SimXpert, SimDesigner, SimManager, Easy5, Sinda, Digimat, Simufact, Actran and MSC Apex. We are committed to the continued development of new CAE technology that integrates disciplines and technologies from standalone CAE tools into unified multi-discipline solvers and user environments. These "next generation" products enable engineers to improve the reliability and accuracy of their virtual prototypes by including multi-physics and multi-discipline interactions. MSC is also the CAE industry's leader in extending simulation to the engineering enterprise. Our customers recognize the need to scale the benefits of virtual prototyping and testing from pockets of experts to mainstream engineering and product development, and MSC offers the only Simulation Data and Process Management platform in the world that has been successfully deployed in industries including automotive, aerospace, shipbuilding, energy, electronics, and more. MSC Software employs 1,200 professionals in 20 countries.

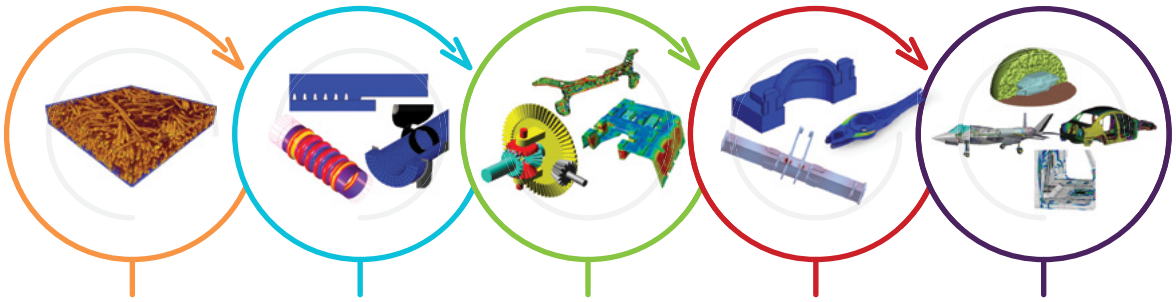
MSC Software makes products that enable engineers to validate and optimize their designs using virtual prototypes. Customers in almost every part of manufacturing use our software to complement, and in some cases even replace the physical prototype "build and test" process that has traditionally been used in product design.



MSC Products

Solution Portfolio

Simulate the Complete Product Engineering Process



MATERIALS	FABRICATION	PARTS	ASSEMBLY	SYSTEMS
Digimat	Digimat Additive	MSC Apex	Simufact Welding	SC/Tetra & scFLOW
Digimat-VA	Simufact Additive	Marc	Simufact Joining	Actran
MaterialCenter	Simufact Forming			MSC Nastran
				Patran
				Adams
				Easy5
				MSC Apex
				SimManager

Instructor-Led Training

MSC Software courses are designed to provide you with exclusive product knowledge. Our course developers and instructors work with product developers gain exposure to new product capabilities. This unique insight is shared with you through official courseware that incorporates the most useful tips and techniques.

Our expert instructors know that not everyone learns in the same way. That's why the MSC training team—engineers themselves with advanced degrees and years of industry experience—uses a variety of techniques to reinforce concepts and build proficiency. These techniques include innovative training materials and hands-on labs that enrich the learning process.

Instructor-Led Training Overview

When it comes to quality instruction in a classroom setting, you can trust MSC Software. Whether you're a student looking to advance your career or start a new one, or a team leader looking to make sense out of new technology, you're virtually guaranteed to find a course that will best suite your training requirements. We offer a variety of Public Courses as well as Custom Courses, held at either an MSC training facility or at the customer's site.

Public Classroom Training

All courses are available in our state-of-the-art classroom facilities and individual computer graphics laboratories at training centres throughout the world. Our courses place heavy emphasis on hands-on computer laboratory work to accelerate practical skills development for all MSC Software products. These courses are typically held in our Newport Beach, CA or Ann Arbor, MI offices.

Training at Your Facility

If you have a number of employees who need training, we offer the cost effective option of bringing our class to your facility. Available worldwide, onsite training is ideal for groups of engineers or those who need customized instruction on MSC products. To maximize productivity in the use of our simulation tools, instructors can tailor the curriculum with company-specific or industry-specific examples, and address challenges and process issues familiar to attendees from your organization.

Customized Courses

If our standard seminar offerings do not meet your training requirements, MSC can develop a course or set of courses tailored to your specific needs. A customized course might include a combination of topics from several standard courses or specialised material not found in any of our standard seminars. MSC will work with your staff to design the course you need.

This Catalog contains a selected range of courses that are offered in the United States. Some courses are already scheduled. For any further queries or to book training please contact us at msc.training@mscsoftware.com. To ensure that we can plan our courses adequately please contact us and register no later than 14 days prior to the planned begin of the course.

Prices mentioned below are per person and a minimum of three attendees are required for a public classroom training to go ahead. Prices for training at your facility or customized courses can be provided on request.

E-Learning Subscription

The MSC Learning Center is for the engineer who quickly needs to be productive with MSC software technologies to simulate product designs, MSC now offers an online e-Learning subscription so that you can get the training you need, anytime you need it!

Designed to address customer needs:

- Shortage of trained and qualified engineers with simulation skills
- Increased access and decreased cost for training
- No travel accommodations or office leave with e-Learning
- Certification Exams for validating simulation skills
- MSC Communities for trainees to collaborate with MSC Experts

Value:

- With access to the e-Learning subscription you get unprecedented access to training with reduced fees and no need to travel or leave the office
- Using the traditional live instructor-led training approach, attending all of the courses in the subscription would cost in excess of $30k and 128 days not including travel
- If you are only interested in one course you still experience incredible savings
  - Average cost of a course is $1,425 and 3 days
- With the e-Learning subscription you get access to 50 courses with over 1000+ hours of training material that is guaranteed to improve your simulation skills

What is Included:

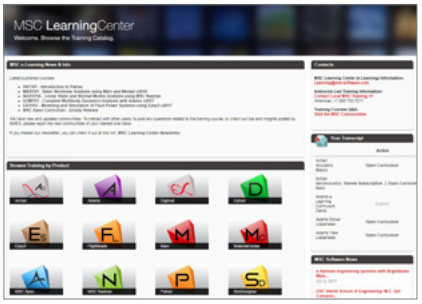
Subscribers to the MSC e-Learning Subscription will receive the following for 1 year from time of purchase:

- Individual named user account for the MSC Learning Center
- Access to all training curriculums for Actran, Adams, Easy5, Patran, Marc, MSC Apex, MSC Nastran and SimManager
  - Online courses are made up several modules
  - Each module contains lecture, demos, audio from subject matter experts, workshops, and review questions that prepare you for the exams
- Certification exams that, upon achieving a passing score, you will receive a certificate for and/or you can post to your public community profile
- Access to the MSC Communities that allow you to:
  - Ask MSC subject matter experts questions about training material
  - Collaborate within your company or with the broader MSC simulation community
- Price $1,500 per named user

Register for a free trial at: https://mscsoftware.csod.com/selfreg/register.aspx?c=elft
More information is available at: http://www.mscsoftware.com/msc-training

Benefits:

- Flexibility, convenience and easy access
- Gain insight and knowledge from MSC experts
- Stay up to date with the latest features in MSC Software Products: Acatran, Adams, Easy5, Patran, Marc, MSC Apex, MSC Nastran and SimManager
- Learn best practices and efficient workflows for common procedures and simulations
- Refresh and/or gain new skill sets that can improve or save time in your day to day simulation and analysis tasks
- Browse through courses and topics of interest to quickly get up to speed to effectively and efficiently streamline your efforts on your project
- Access to the entire curriculum of training courses for Adams, Easy5, Patran, Marc and MSC Nastran
- Collaboration with simulation community through discussion forums
- Certification Exams – prove proficiency in simulation skills and methods and MSC software product proficiency
- Configured Portals – ability to add/ manage custom content



Actran

ACT140 - Launch Acoustics with Actran

Objective

Actran is a finite element tool for vibro acoustic and aero acoustic simulation. This class covers the application of Actran to spacecraft launch acoustics, where an engineer can address the low and mid-frequency range, a concept which is usually not feasible with traditional boundary element solvers. Actran models can be larger, will fun faster, and can be set up to reach higher maximum frequencies than other tools commonly used for this application.

Content

Course topics include the use of the user interface, Actran VI, plus understanding and using the Actran Acoustics and Actran Vibro Acoustics modules, with specific focus on the acoustic effect on the spacecraft resulting from a Turbulent Boundary Layer or Diffuse Sound Field excitation. Outputs will include both structural and acoustic quantities.

Prerequisites Engineering degree, Interest in launch acoustics.

Duration 3 Days
Price $1,425

View Schedule

Adams

ADM701 - Complete Multibody Dynamics Analysis with Adams

Objective

If you're new to mechanical system simulation, this course will give you the foundation of skills you'll need to begin using Adams' powerful virtual prototyping, testing, and visualization capabilities.

Content

- Create parts and organize them into mechanical systems
- Connect parts with ideal constraints, such as:
  - Standard joints (hinges, sliders, etc.)
  - Joint primitives
  - Complex (couplers, screw, etc.)
  - Curve (cam-follower, pin-in-slot)
- Actuate a system with:
  - Ideal and complex part motion
  - Applied forces & gravity
- Connect parts with more realistic forces:
  - Simple (springs, dampers, etc.)
  - Complex based on system states (displacements, velocities, etc.)
  - Complex based on test data
  - Contacts and collisions
- Measure quantities of interest (displacements, velocities, accelerations, applied loads, forces)
- Perform static, transient, and linear simulations
- Precisely control and manage your simulations
- Investigate test results via animations and plots
- Manage files generated and used by Adams/View and Adams/Solver
- "Putting it all together" to complete a comprehensive workshop that tests comprehension of modeling elements and techniques discussed in class
- Additional topics discussed are:
  - Importing CAD-based geometry
  - Performing cam synthesis
  - Applying joint friction
  - Performing design studies
  - Applying sensors to your system

Prerequisites None

Duration 5 Days

Price $2,375

View Schedule

ADM702 - Fundamentals of Multibody Dynamics Analysis with Adams

Objective

This course is intended for users who need a basic understanding of Adams/Solver and Adams/View. This course is a prerequisite for Adams/Car and/or Adams/Chassis training classes. Presented in the class are all of the basics of building models in Adams/View (PARTs, JOINTs, MOTIONs, forces, function expressions, simulation types), running simulations with Adams/Solver and simple plotting with Adams/PostProcessor.

Users who intend to do moderate model creation/optimization in either Adams/View or Adams/Car Template Builder are strongly encouraged to take the 5-day ADM701 Complete Multibody Dynamics Analysis with Adams class instead of this one.

Content

- Building models without a pre-processor
  - Adams dataset structure (.adm)
    - Statements
    - Functions
  - Connecting parts with:
    - Ideal constraints (hinges, sliders, etc.)
    - Flexible connectors (springs, dampers, etc.)
    - Complex forces based on system states or test data
    - Contacts using IMPACT function
  - Actuating a system with:
    - Motions
    - Applied forces
    - Gravity
  - Measuring quantities of interest
    - Displacements
    - Velocities
    - Accelerations
    - Forces
- Simulating models in Adams/Solver
  - Adams command files (.acf)
  - Compare/contrast simulation types
    - Dynamic
    - Kinematic
    - Quasi-static
    - Static
- Reviewing results using the Adams/PostProcessor Adams command files (.acf)

- Plotting/Animating(.msg)
- Message Files (.msg)
- Analysis Files (.gra, .res, .req, .out)

Prerequisites None
Duration 2 Days

Price $ 950

View Schedule

ADM703A - Advanced Modeling Elements and Techniques with Adams/Solver

Objective

This class is intended for moderately experienced Adams users wanting to expand their knowledge of advanced modeling elements and techniques in Adams/Solver. Advanced modeling elements are entities such as the discrete flex View Schedule in Adams/View, the general constraint (GCON) and the differential equation (DIFF) elements. The use of these elements with scripting & function expression logic, new SENSOR functionality and advanced function expressions in Adams/Solver will be discussed.

Content

- Controlling model topology during simulation
- Differential equations for integrating quantities and describing system dynamics
- Implementing simple control algorithms
- Performing standalone Adams/Solver simulations in batch mode.
- Using Adams/Linear as a tool to inspect the system-level linear modes
- Implementing test data via splines
- Advanced simulation logic using SENSORs
- Complex constraints (GCON element)

Prerequisites Complete Multibody Dynamics Analysis with Adams (ADM701) or Fundamentals of Multibody Dynamics Analysis with Adams (ADM702)

Duration 1 Day

Price $475

View Schedule

ADM703B - Adams/Solver Theory: Achieving Robust, Converged Solutions

Objective

This class is intended for experienced Adams users wanting to expand their knowledge of Adams/Solver theory. An emphasis will be put on tying theoretical concepts back to Adams/Solver solution settings (ERROR, HMAX, SI2, MAXIT, etc.). Strategies for creating robust models and sensible solution control settings will be a focus of the course. The various phases of solution (statics, kinematics, dynamics) will be covered in detail and best practices for each will be identified.

Content

- Mathematical descriptions of constraints, body and forces
- Theoretical workings of the static, kinematic and dynamic solvers in Adams
- Numerical Methods for root finding, explicit and implicit integration
- DEBUG/EPRINT interpretation for model debugging
- Error handling strategies for statics and dynamics
- Solver settings for robust static analyses
- Integrator settings for robust dynamics
- Solution convergence techniques
- Modelling best practices

Prerequisites

- ADM701 - Complete Multibody Dynamics Analysis with Adams or ADM702 - Fundamentals of Multibody Dynamics Analysis with Adams
- ADM703A - Advanced Modeling Elements and Techniques with Adams/Solver

Duration 2 Days

Price $950

View Schedule

ADM703C - Writing User Subroutines in Adams/Solver

Objective

This class is intended for experienced Adams users wanting to create Adams/Solver user subroutines. Initial setup with pre-existing libraries is considered, followed by the types of Adams/Solver elements which can be over-ridden. Dealing with user input is considered along with usage of the many built-in utility subroutines. Querying Adams/Solver for system state information (displacements, velocities, forces, etc.) is covered in detail, followed by initialization (IFLAG) and differencing (DFLAG) considerations. Callback subroutine functionality is presented for easy detection of simulation events (convergence, simulation start/end, saving/reloading, etc.) and considerations for threadsafe computations are discussed.

Students are expected to be comfortable with simple programming tasks in Fortran; this class does not provide a formal nor comprehensive introduction to the Fortran programming language.

Content

- Using existing Adams/Solver libraries
- Element types to over-ride, what is to be calculated
- Inputs using the PAR array
- Interface differences between Fortran and C
- Querying system states using utility subroutines
- Handling initialization and differencing calculations
- Call back subroutines
- Threadsafe computation considerations

Prerequisites

- Moderate knowledge of the Fortran programming language
- ADM701 - Complete Multibody Dynamics Analysis with Adams or ADM702 - Fundamentals of Multibody Dynamics Analysis with Adams
- ADM703A - Advanced Modeling Elements and Techniques with Adams/Solver
- ADM703B - Adams/Solver Theory: Achieving Robust, Converged Solutions

Duration 1 Day

Price $475

View Schedule

ADM704A - Advanced Parametrics, Design Sensitivity, and Optimization using Adams/View

Objective

This course is intended for users who want parameterize their model in order to determine how different modeling parameters influence the design and how to iterate on those to achieve the optimal design using Design Studies, Design of Experiments (DOE) and Optimization capabilities in Adams/View.

Content

- Introduction to parameterization.
- Understanding expressions used in a parameterized model.
  - Built-in ADAMS/View functions (location, orientation, modeling, etc.).
  - Accessing database objects and their component values.
- Creating a parametric model using:
  - Construction Points
  - Design Variables
  - Expression Builder
  - Graphical Tools
  - Table Editor
- Performing multi-run simulations to better understand and improve overall system behavior
  - Design Study
  - Design of Experiments (DOE)
  - Optimization
- Creating design objectives and design
- Constraints for your system. Interpreting results of a multi-run simulation.
  - Plotting
  - Summary Tables

Prerequisites ADM701 (Complete Multibody Dynamics Analysis with Adams)

Duration 2 Days

Price $950

View Schedule

ADM704B - Automating Tasks using Adams/View Scripting, Macros, and GUI Customization

Objective

This course focuses on automating tasks in Adams/View for efficiency. The Adams/View Command Language is studied in detail along with looping, macros and the creation of custom menus and dialog boxes.

Content

- ADAMS/View command language and structure.
  - Commands – keywords, parameter names, parameter values
  - Conditional constructs (if/then/else, for loops, etc.)
  - Using command files to perform tasks
- The macro language and macro structure.
- How to create and access macros to automate common tasks.
- Creating custom dialog boxes
  - Using the dialog box builder
  - Defining labels, fields, buttons, options menu, etc.
  - Become familiar with underlying commands
- Creating custom menus.
- Putting it all together to build a custom interface.

Prerequisites ADM701 (Complete Multibody Dynamics Analysis with Adams) and ADM704A (Advanced Parametrics, Design Sensitivity, and Optimization using Adams/View).

Duration 2 Days

Price $950

View Schedule

ADM710 - Flex Body Dynamics and Modal Stress Recovery using Adams

Objective

You will learn how to use Adams/Flex to incorporate flexibility into your Adams models. The course is primarily focused on using component modal synthesis via the Modal Neutral File (MNF) and Adams/View. Previous FEA experience is a plus, but not a requirement. You will be comfortable using Adams/Flex after taking this course.

Content

- Modal Flexibility: Theoretical background, combined rigid-flexible equations of motion
- Importing a flexible body into a rigid-body model
- Working with flexible body models in Adams/View (mode manager, connect flexible part to rigid parts, add forces/torques, use measures)
- Adams/Solver dataset and Adams/View command language syntax
- Using flexible body models with stand-alone Adams/Solver (edit .adm files, generate .mtx files, submit simulations)
- Performing static, dynamic, and eigenvalue analyses
- Exporting loads from Adams/View
- Methods for optimizing MNFs
- Comparing results of rigid and flexible models (plotting, animation)
- Validating Adams/Flex results with FEM results and hand-calculations
- Modeling considerations
- Debugging
- Modeling contact with a flexible body
- Using a modal force (MFORCE) to define a pressure load
- Publishing simulation results on the web (generate movies, table of eigenvalues, snapshot images, include model files)
- Reviewing several application examples
- Reviewing/Discussing methods for generating MNF from FEA packages
- Adams/AutoFlex

Prerequisites

- Adams Full Similation users: Basic Adams Full Simulation Course
- Adams/Car users: Adams/Car Course

Duration 3 Days

Price $1,425

View Schedule

ADM711 - Control System Integration with Adams using MATLAB and Easy5

Objective

You will learn how to connect your Adams models to control systems developed in MATLAB or Easy5. Techniques for combining linear, nonlinear, continuous, and sampled control systems with your Adams model will be presented, along with tutorials. Converting your MATLAB or Easy5 model into a native Adams entity via Control System Import to run the combined model completely within Adams will also be discussed. Other topics presented in the course include an overview of all System Elements, including State Variables, Differential Equations, Linear State Equations, and General State Equations to develop models (e.g., control systems) supplemental to your mechanical model.

Content

- Adams/Controls terminology
- Defining inputs/outputs for Adams models to tie to controls
- Adams Interface elements provided by Adams/Controls for MATLAB, and Easy5
- Function evaluation mode and co-simulation mode
- Interactive and batch mode simulations
- using initialization commands
- Control system examples with differential equations, transfer functions, linear state equations, and general state equations
- Examples using Control System Import with Easy5 and MATLAB

Prerequisites Basic Adams Full Simulation Course or equivalent experience with Adams and MATLAB - SimuView Schedule, or EASY5 experience preferred.

Duration 2 Days

Price $950

View Schedule

ADM720 - Frequency Domain Analysis using Adams/Vibration

Objective

This course is intended for users who want to perform frequency-domain analysis. Using Adams/Vibration, you can study forced vibrations within the Adams model at isolated instances. The results from Adams/Vibration can be used in noise/vibration/harshness (NVH) studies.

Content

- Defining the input channels from which you will actuate (shake) the model: Kinematics (Disp., Vel., and Accel.), Force, State Variables.
- Working through examples that demonstrate each available option for actuating the system: Swept sine, PSD, Rotating Mass, User-defined.
- Defining the output channels from which to analyze the simulation.
- Performing vibration analysis in batch mode using Python.
- Vibration analysis of modeling containing flexible bodies.
- using Desing Evaluation to minimize frequency response
- Brief section on theory

Prerequisites Adams Full Simulation Users: ADM701 (Complete Multibody Dynamics Analysis with Adams) required and ADM704A (Advanced Parametrics, Design Sensitivity, and Optimization using Adams/View) and ADM704B (Automating Tasks using Adams/View Scripting, Macros, and GUI Customization) recommended. Adams/Car Users: ADM701 (Complete Multibody Dynamics Analysis with Adams) required.

Duration 1 Day

Price $475

View Schedule

ADM730 - Design of Experiments (DOE) and Stochastics (Monte Carlo) Analysis using Adams

Objective

After taking this course you will be able to:

- Create Factors and Responses in your model
- Understand how Design Variable range settings work
- Create complex Response definitions in Adams
- Understand DOE Screening/Response Surface Strategies
- Be comfortable with DOE output statistics
- Create Monte Carlo variation studies
- Effectively create and run large design variation studies

Content

- Factors and Responses
- Parametrics
- Classical DOEs
- Results Interpretation
- Using the ASCII Conduit
- EASY5 and Adams/Insight
- Monte Carlo Analysis, Correlation

Prerequisites ADM701 (Complete Multibody Dynamics Analysis with Adams) or equivalent

Duration 1 Day

Price $475

View Schedule

ADM740 - Vehicle Modeling and Simulation using Adams/Car

Objective

Experienced Adams user can learn how to automate the process of performing general and customized vehicle simulations. This is taught using built-in tools that allow for the creation of fully parameterized and modularized vehicle models in the Adams/Car environment.

Content

- Description about user modes and data hierarchy
- Overview of database hierarchy
- Learning the concept of subsystems and assemblies
- Create new suspension and full vehicle models
- Create user specific events with the Driving Machine
- Setup suspension and full vehicle simulations
- How to use handling and durability tire
- Perform suspension and full vehicle simulations
- Create plot configuration files
- Good practice in creating a fully parametric vehicle model
- Create and modify templates
- Create and modify different type of parts
- Learn how to use communicators
- Learn how to create new requests within the same subsystem as well as between different subsystems
- Create and modify different property files
- Create and modify automotive elements such as spring, dampers, bushings, bumpstop, etc.
- Overview on how to implement flexible bodies

Prerequisites

- Basic Adams Full Simulation Course
- General vehicle knowledge (vehicle dynamics, suspension design, etc.) preferred

Duration 4 Days

Price $1,900

View Schedule

ADM741 - Vehicle Modeling and Simulation using Adams/Driveline

Objective

This course is intended for users who need to create assemblies of suspensions and full vehicles, including driveline components, and then analyze them to understand their performance and behavior. Presented in the class are all of the basics of building models in Adams/Driveline (engine, gearbox, prop shafts, and differentials), running simulations and simple plotting with Adams/PostProcessor.

Content

- Understanding features of Adams Driveline
- Modeling Components in Adams Driveline
  - Understanding different components- Gear Forces, Gear Pairs, Gearbox Assembly, Gearbox Assembly, Gearbox Assembly, Differential , Differential-Limited Slip Diff, Torque Converter, Flexible Connection Components, other Modeling Components
  - Exercise on Modeling Gearbox and Driveline components
- Driveline Analysis
  - Driveline Analyses – Understanding different analyses in Adams /Driveline
  - Setting up Initial-Velocity Analysis
  - Specific Test analysis – Running analyses and Post-processing and investigating results in Adams/Post-Processor
- Full Vehicle Analysis
  - Communicators - setup the powertrain/driveline model for a full vehicle
  - Preparation for Quasi-Statics- understanding requirements that need to be fulfilled in order to support Quasi-Static setup analysis
  - Performing full vehicle analysis
  - Post-processing and investigating results in Adams/Post-Processor

Prerequisites ADM740 - Vehicle Modeling and Simulation using Adams/Car

Duration 1 Day

Price $475

View Schedule

ADM750 - Gear, Belt, and Chain Modeling with Adams/Machinery

Objective

For users wanting to build detailed models containing belts, chains or gears with Adams/Machinery.

Content

- Introducing the Adams/Machinery Wizard
- Adams/Machinery Database Structure for Components, Parametrics, etc.
- Creating Gear Systems:
  - Spur, Helical & Bevel Gear Types
  - Simplified, Detailed & 3D Contact Types
- Creating Belt Systems:
  - Smooth, V-Grooved & Toothed Belts
  - Pulley Types: Standard and Tensioners
  - Actuator Types and System Outputs
- Creating Chain Drive Systems:
  - Roller and Silent Types
  - Compliance Methods: From Linear to User-Defined
  - Tensioner/Guide Types: Rotational, Fixed and Translational
- Simulation Speed and Robustness Guidelines for Typical Machinery Systems

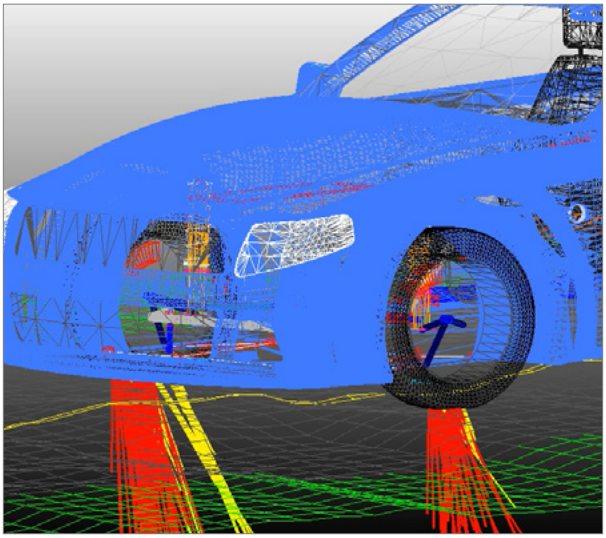
Prerequisites ADM701 (Complete Multibody Dynamics Analysis with Adams).

Duration 1 Day

Price $475

View Schedule

ADM761 - Basic Suspension and Full Vehicle Analysis using Adams/Chassis



Objective

For users wanting to build detailed models containing belts, chains or gears with Adams/Machinery.

Content

- Description and usage of Adams related products
  - Adams/Solver
  - Adams/PostProcessor
  - Adams/Insight
- Description of Chassis model file structure
  - Systems
  - Subsystems
  - Property Files
  - Database Structure
- Investigation of half-vehicle model behavior
  - Using the Suspension Design Module (SDM) requests
  - Bushing definition and orientation techniques
  - Attachment options
  - Hub compliance modeling
  - Effect of vehicle loading on steering behavior
  - Using the static load case event
- Investigation of full-vehicle model behavior
  - Effect of passengers on steering response
  - Understanding understeer budget contribution
  - Using the Static Vehicle Characteristics (SVC) event
  - Stringpots, accelerometers, and ride height sensors to 'instrument' your model
  - Simple chassis stiffness modeling
  - Brake system modeling overview
  - Trailer modeling
  - Introduction to Chassis customization
  - Converting files from Adams/Pre to Adams/Chassis

Prerequisites ADM701 (Complete Multibody Dynamics Analysis with Adams) or ADM702 (Fundamentals of Multibody Dynamics Analysis with Adams).

Duration 2 Days

Price $950

View Schedule

Digimat

DIG 101 - Digimat Introductory Training - Structural Engineering

Objective

This Introductory course is geared to all engineers using composite materials for FE structural Analyses.

Content

- Digimat-MF & Digimat-MX - Definition and calibration of a Digimat material
- Digimat-CAE - Use of a Digimat Material in Marc / Nastran. Capture the anisotropic and non-linear behavior of the material
- Digimat-MAP - Mapping of fiber orientation from the injection to the structural meshes

Prerequisites Basic theoretical knowledge of material constitutive models (elasticity, elasto-plasticity) is advised.

Duration 1 Day

Price $475

View Schedule

DIG 102 - Digimat Introductory Training - Material Engineering

Objective

Products: Digimat-MF, Digimat-MX, Digimat-FE
This Introductory course is geared to all engineers who would like to learn more about modeling and predicting the behavior of composite materials.

Content

- Digimat-MF & Digimat-FE - Validation of new material design
- Digimat-MF & Digimat-MX - Material design optimization to reach specifics targets

Prerequisites Basic theoretical knowledge of material constitutive models (elasticity, elasto-plasticity) is advised.

Duration 1 Day

Price $475

View Schedule

DIG 103 - Digimat Introductory Training - Material & Structural Engineering

Objective

Products: Digimat-MF, Digimat-MX, Digimat-MAP, Digimat-CAE
This Introductory course is geared to all engineers using composite materials for FE structural Analyses.

Content

- Digimat-MF - Definition of a Digimat material
- Digimat-MX - Material database management
- Digimat-CAE - Use of Digimat Material in Marc / Nastran. Capture the anisotropic of the material
- Digimat-MAP - Mapping of fiber orientation from the injection to the structural meshes
- Digimat-FE - Introduction to designing RVE (representative volume element) for various composite microstructures

Prerequisites Basic theoretical knowledge of material constitutive models (elasticity, elasto-plasticity) is advised.

Duration 1 Day

Price $475

View Schedule

DIG 120 - Digimat Standard Training - Material & Structural Engineering

Objective

Products: Digimat-MF, Digimat-MX, Digimat-FE
This course will teach the student how to use the Digimat products to virtually elevate the performance of new composite material design and to apply this Digimat material model in their FE Analysis of a composite part that has fiber orientations predicted by an injection modeling or draping simulation.
Who should attend: Engineers involved in the design of new composite materials or/and in the structural analysis of parts using composite materials.

Content

- Introduction to the Digimat Graphical User Interface
- Mean-field homogenization theory as well as the material models available in Digimat-MF
- Material Database management and automatic calibration of Digimat materials in Digimat-MX
- Coupling of Digimat with FEA code in Digimat-CAE. Introduction to the micro solution procedure and brief

presentation of the failure criteria definition

- Mapping of fiber orientation tensors, initial stresses/ temperatures between two dissimilar meshes in Digimat-Mat.
- Introduction to RVE (representative volume element) for two phases composite microstructures using Digimat-FE
- Exporting the RVE into Nastran or Marc to run a finite element analysis on the material and post-processing of the results in Digimat-FE
- Guided exercises to implement the different procedures presented during the course

Prerequisites Basic theoretical knowledge of material constitutive models (elasticity, elasto-plasticity) is advised. A good knowledge on MSC Nastran or Marc is welcome.

Duration 2 Days

Price $950

View Schedule

DIG201 - Digimat Standard Training - Structural Engineering

Objective

This course will teach the student how to use a Digimat material model in their FE Analysis of a composite part that has fiber orientations predicted by an injection modeling or draping simulation. By using a Digimat material model each student will then take into account the non-linear material properties of their composite and effects of local microstructure on their part.
Who should attend: Engineers involved in the design or structural analysis of parts using composite material.

Content

- Introduction to the Digimat Graphical User Interface including the plug-in embedded CAE codes
- Mean-field homogenization theory as well as the material models available in Digimat-MF
- Material Database management and automatic calibration of Digimat materials
- Coupling of Digimat with FEA code in Digimat-CAE. Introduction to the different solution procedure (macro, hybrid, micro) and presentation of the failure criteria definition
- Mapping of fiber orientation tensors, initial stresses/ temperatures between two dissimilar meshes in Digimat-Mat. Theoretical aspects of the mapping procedure are explained
- Guided exercises to implement the different procedures presented during the course

Prerequisites Basic theoretical knowledge of material constitutive models (elasticity, elasto-plasticity) is advised. Good Knowledge of one FEA code is mandatory.

Duration 2 Days

Price $950

View Schedule

DIG202 - Digimat Standard Training – Material Engineering

Objective

Products: Digimat-MF, Digimat-MX, Digimat-FE
This course will teach the student how to use Digimat products to virtually elevate the performance of new composite material design.

Content

- Introduction to the Digimat Graphical User Interface
- Mean-field homogenization theory as well as the material models available in Digimat-MF
- Material Database management and automatic calibration of Digimat materials in Digimat-MX
- Introduction to designing RVE (representation volume element) for various composite microstructures including composites with; multiple reinforcing phases, clustering, and dechesion zones, among others, using Digimat-FE
- Exporting the RVE into Marc / Nastran to run a finite element analysis of the material and post-processing of the results in Digimat-FE
- Guided exercises to implement the different procedures presented during the course

Prerequisites
Basic theoretical knowledge of material constitutive models (elasticity, elasto-plasticity) is advised. A good knowledge on MSC Nastran or Marc is welcome.

Duration 2 Days

Price $950

View Schedule

DIG301 - Digimat Advanced Training - Aerospace Application

Objective

Products: Digimat-MF, Digimat-MX, Digimat-FE
This advanced course provides each student with training on the latest technology implemented in Digimat to better predict the

thermo-mechanical behavior of their composite part in isothermal and non-isothermal stiffness analysis and impact analysis.
Who should attend: Engineers involved in the design of aerospace parts.
Material: woven, HexMC, laminate.

Content

- Calibration of Digimat materials to predict the behavior of composite material for thermo-mechanical stiffness analysis and impact study
- Failure criteria definition using the first psuedo grain method with the micro and hybrid solution procedures to predict progressive failure of the part
- Take advantage of the micro/macro results computed by Digimat during the FEA to further optimize designs.
- Optimize your Digimat material model to decrease computation time
- Exercises to implement the different procedures presented during the course
- Definition of the best strategy to integrate Digimat technology into the design process

Prerequisites Basic experience with Digimat is required. Good knowledge of at least one FEA code is mandatory.

Duration 2 Days

Price $950

View Schedule

DIG325 - Digimat Advanced Training - Electrical/Electronic Application

Objective

Products: Digimat-MF, Digimat-MX, Digimat-FE
This advanced course provides each student with training on the latest technology implemented in Digimat to better predict the thermo-mechanical behavior of their electronics part in isothermal and non-isothermal stiffness analysis. Electronics performance of nano-composite material is also investigated by using Finite Element homogenization.
Who should attend: Engineers involved in the design of electric/electronic parts. Material: shrot fiber plastic reinforced, chopped fiber, nano-composite.

Content

- Calibration of advanced Digimat materials for thermo-mechanical
- Optimizing the Digimat material model to decrease computation time
- Learn how to better take advantage of the micro/macro results computed by Digimat during the FEA to futher optimize designs
- In depth study of nano-composite electrical and mechanical performance
- Exercises to implement the different procedures presented during the course
- Definition of the best strategy to integrate Digimat technology inside the design process

Prerequisites Basic experience with Digimat is required. Good knowledge of at least one FEA code is mandatory.

Duration 2 Days

Price $950

View Schedule

DIG340 - Digimat Advanced Training - Automotive Application

Objective

Products: Digimat-MF, Digimat-MX, Digimat-FE
This advanced course provides each student with training on the latest technology implemented in Digimat to better predict the thermo-mechanical behavior of their composite part in isothermal and non-isothermal stiffness analysis, crash analysis and fatigue analysis.
Who should attend: Engineers involved in the design of automotive parts.
Material: short fiber plastic reinforced, chopped fiber.

Content

- Callabration of advanced Digimat materials for thermo-mechanical and crash analysis (Thermo-ViscoElastic, Thermo-elastoviscoplastic, etc.)
- Failure criteria definition using the first pseudo grain method with the micro and hybrid solution procedures
- Optimizing Digimat material model to decrease computation time
- Learn how to better take advantage of the micro/macro results computed by Digimat during he FEA to futher optimize designs
- Exercises to implement the different procedures presented during the course
- Definition of the best strategy to integrate Digimat technology inside the design process

Prerequisites Basic experience with Digimat is required. Good knowledge of at least one FEA code is mandatory.

Duration 2 Days

Price $950

View Schedule

Dytran

DYT101 - Introduction To Dytran

Objective

Dytran is designed to solve transient dynamic problems involving a high degree of nonlinearity. This seminar provides an introduction to the Lagrangian capabilities of Dytran. The primary emphasis is on how to use the program to solve engineering problems. The major capabilities of the program are covered in detail. The process of performing an analysis is discussed in its entirety, from initial modeling to the post processing of results.

Hands-on workshops and example problems reinforce the material covered in the lectures. Advice is offered on modeling techniques, meshing, and evaluation of results. In addition, techniques to minimize the cost of analyses are discussed. By the end of the seminar, attendees should be able to apply Dytran to the solution of practical engineering problems in structural mechanics.

Content

Day One

- Introduction
  - Overview of Dytran capabilities
  - Differences between Lagrangian and Eulerian technology
  - Overview of contact and Euler/Lagrange coupling techniques
  - Typical applications
- Explicit transient dynamic analysis
  - Introduction to explicit solution techniques
  - Explicit versus implicit technology
  - When to use explicit technology
- Input definition
  - Overview of the input file
  - Input file formats and data generation
  - File management system
  - Executive control
  - Case control
  - Bulk data
- Running the analysis
  - Modeling
  - Description of the Dytran files
  - Example input file
  - Restarts and rezones
  - Executing Dytran
  - Postprocessing using the XDEXTR translator
- Workshop on using XDEXTR
- Basic concepts of Lagrange
  - Theory
  - Computational cycle
  - User subroutine implementation

Day Two

- Lagrangian capabilities
  - Element library
  - Material models
  - Loads and constraints
  - User subroutines
- Workshop example on Lagrange
- Pre-stressing with MSC Nastran
- Concept of surfaces in Dytran
  - Surface definition
  - Defining segments
  - Surface modeling
- Lagrangian Interaction Capabilities
  - Contact
  - Tied connections
  - Kinematic connections
  - Breakable connections
- Modeling techniques
  - Mesh design
  - Problem simplification
  - Postprocessing
  - Results interpretation
- Workshop example on Lagrangian interaction

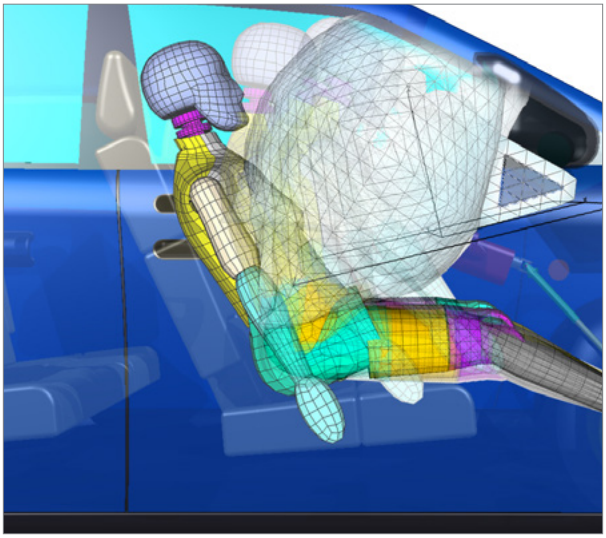
Prerequisites Experience with a general-purpose finite element analysis application is recommended

Duration 2 Days

Price $950

View Schedule

DYT103 - Introduction to Airbag Analysis and Occupant Safety using Dytran



Objective

In this course we will discuss the steps to perform an Occupant Safety Analysis. Some of the topics to be covered are: discussion on the input cards needed, modeling of an airbag, self contact, inflator, holes, and permeability and heat losses. Patran will be used to position a Dummy, define contact between Dummy and airbag and contact between Dummy and seat belt. Several workshops are used to illustrate these techniques, followed by discussion of the results for a more thorough understanding of the problems analyzed.

Content

- Air bag Modelling in Uniform Pressure and CFD coding
- Definition of holes, permeability, heat loss, inflator characteristics and mixture of gases
- Air bag Self Contact
- ATB Dummy Positioning
- Occupant Dummy Air Bag Interaction

Prerequisites

Basic knowledge of the Dytran software is required. Basic knowledge of Patran and the Dytran Preference is required. Basic knowledge of Occupant Safety is not required but will be beneficial.

Duration 2 Days

Price $950

View Schedule

Easy5

EAS101 - Dynamic System Modeling and Simulation using Easy5

Objective

This course gives the engineer a basic understanding of the Easy5 modeling, simulation, and analysis environment. Students learn to represent an existing dynamic system on screen using predefined modeling blocks and components, empirical table data, and user-defined Fortran components. Students learn to calculate stable operating points using the Easy5 Steady-State Analysis, and are introduced to fixed- and variable-step integration used during simulation. The class also covers the Easy5 set of linear analyses, including, in part, transfer function calculation, frequency response generation, and root locus analysis.

Content

- Model Building
- Linear Analysis
- Analysis Tool & Methodology
- Simulation and Integration
- Data Tables and the Matrix Editor
- Operating Point and Steady State Analysis
- Easy5 Architecture
- Writing Code in Easy5

Prerequisites An engineering background, knowledge of FORTRAN, and an understanding of differential equations.

Duration 2 Days

Price $950

View Schedule

EAS103 - Modeling and Simulation of Fluid Power Systems using Easy5

Objective

This course has been designed to give the hydraulics engineer a basic understanding of the issues and difficulties surrounding modeling and simulation of fluid power systems. It is geared towards the user who already has a general understanding of the Easy5 modeling, simulation and analysis environment as well as knowledge in the area of hydraulics design.

Students will learn about the specific issues that arise when building and analyzing a fluid power model. They will gain hands-on experience by building a number of different systems using predefined hydraulics and valve kit components as well as incorporating general purpose components and user-defined Fortran code. Models will include components such as valves including valve dynamics, pumps, circuits with multiple loops and heat exchangers. Open and closed loop systems will be built.

The course will cover the importance of calculating steady state conditions and provide tips and insights into how to do this. Simulation and transient water hammer effects will be discussed as well. Students will learn how to include temperature considerations in their models and how to use Easy5 components to parameterize their models.

Content

- General Theory of Hydraulic Modelling in Easy5
- Modelling an Open Loop Oil Cooling System
  - Obtain Initial Operating Points
  - Fluid properties
- Modelling a Closed Loop Oil Cooling System
  - Difficulties in obtaining steady state
- Building a Piloted Servo Valve
  - Building valves from primitive hydraulic components
  - Use steady state scan to parameterize models
  - Linear analysis
- Model a Raise/Lower Valve for a Hydraulic Lift
  - Using HC library components to create larger component
  - Reverse flow in a hydraulic system
  - Minimizing number of pressure states
- Simulating Water hammer Effects
- Disaster Recovery - How to handle problem models

Prerequisites Working knowledge of Easy5 and a basic understanding of the Thermal Hydraulics library.

Duration 2 Days

Price $950

View Schedule

EAS105 - Modeling and Simulation of Gas Systems using Easy5

Objective

This course will teach you how to use Easy5 to model pneumatic systems and valves. It will review some fundamentals that are usually not well understood and provide advanced instructions for features in Easy5.

Content

- General Theory of Pneumatic Modelling in Easy5
- Modelling a Simple Pneumatic Pressure Regulator
  - Practice basic Easy5 skills
  - Obtain Initial Operating Points
  - Difficulties in Obtaining Steady State
  - Use Steady State to parameterize models
- Modelling a Flow Control Valve
  - Building Valves from Primitive Pneumatic Components
- Building an Electro pneumatic Pressure Regulator
  - Data Tables and the Matrix Editor
- Linear Analysis
- Model a Temperature Control System with Heat Exchangers
- Simulation and Numerical Integration
- Macro Code Development
- Sorting and Solving Implicit Loops
- Modelling and Development with Discrete Components
- Modelling Discontinuities with Switch States
- Additional topics as interest and time allows including:
  - Debugging models
  - How to define and use your own fluid property set

Prerequisites Completion of introductory class EAS101 (Dynamic System Modeling and Simulation using Easy5) and/or a working knowledge of Easy5.

Duration 2 Days

Price $950

View Schedule

EAS107 - Modeling and Simulation of Multi-Phase Fluids using Easy5

Objective

This course will provide the student with an understanding of how to model closed-loop, multi-phase systems using Easy5. Applications include HVAC, environmental controls, and refrigeration. The governing physical principles and the corresponding equations of state will be presented as well as a detailed example model.

Content

- Overview of the Mulitphase (VC) library
- Governing dynamic equations
- Phase and phase change modeling
- Orifice flow
- Using published fluid data tables

Workshops include:

1. Refrigerant line
2. Adding a compressor
3. Modeling a condenser and expansion valve
4. Adding an evaporator
5. Closing the refrigerant loop
6. Adding EEV control
7. Heat exchange to the environment
8. Calculating COP and a stable closed-loop operating point
9. Thermostatic control and realistic environmental conditions
10. Generating custom fluid properties for the Multiphase library

Prerequisites EAS101 (Introduction to Dynamic System Modeling and Simulation using Easy5) or equivalent experience.

Duration 2 Days

Price $950

View Schedule

Flightloads

FLD120 – Aeroelasticity using FlightLoads and Patran

Objective

This seminar is intended for engineers concerned with structural loads, flying qualities, and aeroelastic stability of flexible aircraft and missiles. The objective of the seminar is to familiarize the engineer with an integrated approach to the state-of-the-art MSC Nastran applications in aeroelastic analyses and their implementation via the FlightLoads User Interface and process management tool. An overview of the aeroelastic capability is followed by discussion of the available aerodynamic theories and case studies. Highly detailed workshops are used throughout.

Content

- Introduction to Aeroelasticity
- Introduction to FlightLoads and Dynamics
- Aerodynamic Theories
  - Doublet Lattice
  - Zona51
  - Subsonic Slender Body Theory
- Surface Splines
- Linear Splines
- Theory of Static Aeroelastic Analysis
- Monitor Points
- Flutter Analysis
- Aeroelastic Response Analysis
- Basics of Patran and MSC Nastran
  - Patran Graphical User Interface
  - MSC Nastran Files
- Mapping CFD Data to a Structural FE Model
- Load Mapping

Prerequisites

NAS101A and B (Linear Analysis using MSC Nastran)

NAS102A and B (Dynamic Analysis Using MSC Nastran)

PAT301 (Introduction to Patran)

Duration 5 Days

Price $2,375

View Schedule

FLD120S – FlightLoads and Aeroelasticity - Static Analysis

Objective

This seminar is intended for engineers concerned with structural loads, flying qualities, and aeroelastic stability of flexible aircraft and missiles. The objective of the seminar is to familiarize the engineer with an integrated approach to the state-of-the-art MSC Nastran applications in aeroelastic analyses and their implementation via the FlightLoads User Interface and process management tool.

An overview of the aeroelastic capability is followed by discussion of the available aerodynamic theories and case studies of static aeroelasticity. Highly detailed workshops are used throughout to exercise the student and ensure the lessons are well learned.

Content

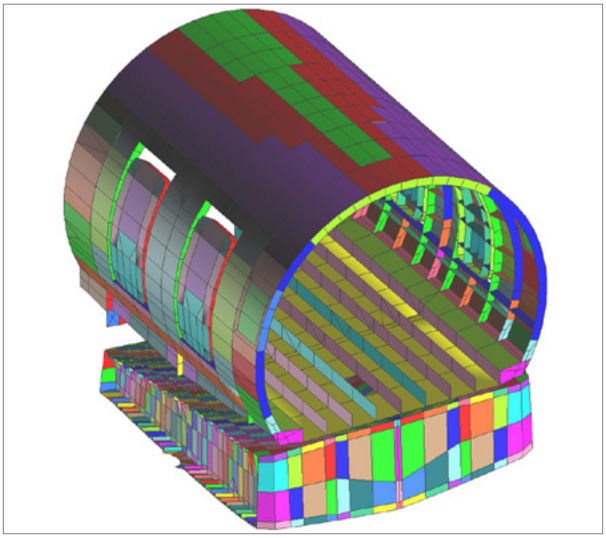
- Introduction MSC Nastran Aerodynamics
- Introduction MSC Nastran Splining
- Introduction MSC Nastran Aeroelastic Solutions
  - Static Aeroelasticity
- Introduction to FlightLoads
- Aerodynamic Theories
  - Doublet Lattice
  - CPM
  - Zona51
  - Slender Bodies
- Aeroelastic Overview
  - Surface Splines
  - Linear Splines
- Basics of Patran and MSC Nastran
  - Patran GUI
  - Nastran File Format
  - Nastran Syntax
  - Static Aeroelasticity
- Theory
  - Case Study 1 and Workshop - FSW aircraft in longitudinal trim
  - Case Study 2 and Workshop - Longitudinal Analysis of a straight wing aircraft
  - Case Study 3 and Workshop - Aileron Effectiveness in Roll
  - Case Study 4 and Workshop - Advanced fighter aircraft analysis
  - Case Study 5 and Workshop - Advanced Wing and Canard Aero Workshop

Prerequisites
NAS101A (Linear Static and Normal Modes Analysis using MSC Nastran)
NAS 101B (Advanced Linear Analysis using MSC Nastran)
NAS120 (Linear Static Analysis Using MSC Nastran and Patran) or equivalent
NAS102A (Dynamic Analysis using MSC Nastran)
NAS102B (Advanced Dynamic Analysis Using MSC Nastran)
PAT301 (Introduction to Patran)

Duration 3 Days

Price $1,425

View Schedule



Marc

MAR101 – Basic Nonlinear Analysis using Marc and Mentat

Objective

The purpose of this course is to introduce the new Marc user to both Marc and Mentat by lectures and hands on modeling of nonlinear problems.

Content

- Introduction to Mentat
- Nonlinear Finite Element Analysis
  - Geometrically Nonlinear Analysis
  - Material Nonlinear Analysis
  - Modeling with Contact
- Resolving Convergence Problems
- Numerical Analysis of Nonlinear Problems

Prerequisites A basic knowledge of statics and strength of materials is highly recommended. Previous finite element analysis experience is recommended.

Duration 3 Days

Price $1,425

View Schedule

MAR102 - Advanced Nonlinear Analysis using Marc and Mentat

Objective

The purpose of this course is to enhance the current Marc user's understanding of modeling nonlinear problems. Lectures are supported by hands-on modeling of nonlinear problems.

Content

- Material Nonlinearity
- Contact
- Adaptive Meshing
- User Subroutines in Marc
- Heat Transfer and Thermal Stresses
- Global - Local (Structural Zooming) Analysis in Marc
- Restarts
- Performance
- Workshop Problems
  - Experimental Curve Fitting Using Physical Test Data
  - Creep of Tube
  - Superplastic Forming of a Metal Container
  - Composite Progressive Failure Analysis using VCCT
  - Elastomeric Cylinder (Segment to Segment Contact)
  - Ship Bumper Contact Analysis
  - Global Remeshing
  - Local Adaptive Remeshing
  - Creep of a Tube (User Subroutine)
  - Heat Transfer (Conduction and Convection)
  - Heat Transfer (Radiation)
  - Coupled Analysis (Thermal / Structural)
  - Global – Local (Structural Zooming)
  - Restarts

Prerequisites A basic knowledge of nonlinear simulations - Familiarity with Mentat 2011 - Completion of MAR101 (Basic Nonlinear Analysis using Marc and Mentat) or equivalent experience

Duration 3 Days

Price $1,425

View Schedule

MAR103 – Elastomer Analysis using Marc and Mentat

Objective

The purpose of this course is to provide a fundamental understanding of how material testing and finite element analysis are combined to improve the design of rubber and elastomeric products.

Content

- Introduction
- Overview of Elastomer Testing and Analysis
  - Test data are dependent on the measurement method
  - Analysis results are dependent on the mesh
  - Measurement and Modeling principals
- Uniaxial Tension/Compression Testing and Analysis
  - Specimen setup and test
  - Set up model - Curve Fitting of Uniaxial Material Data
  - Run Simulation
  - Understand Physical and Numerical Results

- Biaxial Tension/Compression Testing
  - Specimen setup and test
  - Set up model - Curve Fitting of Multi Mode Material Data
  - Run Simulation
  - Understand Physical and Numerical Results
- Pure Shear Testing
  - Specimen setup and test
  - Set up model - Curve Fitting of Multi Mode Material Data
  - Run Simulation
  - Understand Physical and Numerical Results Contact Analysis
- Product Simulations with Specimen Data
  - Definition of contact bodies
  - Contact and friction
  - Case Histories of Product Simulations

Prerequisites A basic knowledge of statics and strength of materials is highly recommended. Previous finite element analysis experience is recommended. Knowledge of elastomeric materials.

Duration 3 Days

Price $1,425

View Schedule

MAR106 – Electromagnetic Analysis using Marc and Mentat

Objective

This course provides an overview of general electromagnetic theory and of the theory behind different analysis types in Marc Electromagnetics and the typical problems they can handle. It provides a quick review of Marc nonlinear methodology and contact analysis as well as of Marc Structural and thermal analysis. For each analysis type, the workshops are chosen to show a range of problems that can be solved in Marc. Each workshop shows detailed step by step finite element modeling in Mentat and is a quick, simple and efficient way of learning Mentat. The post-processing section of each workshop problem illustrates how finite element results can be interpreted, and how they can be used to obtain other practical quantities. Relevant short notes at the end of a workshop help in getting additional information about Marc and Mentat.

Content

- Day 1
  - Main theory of EM and Electrostatic workshop
- Day 2
  - Joule-thermal-structural and Magnetostatics structural workshop
- Day 3
  - Piezoelectric , magnetostatic-thermal and magnetodynamics workshop

Prerequisites None

Duration 3 Days

Price $1,425

View Schedule

MAR120 - Basic Nonlinear Analysis using Marc and Patran

Objective

MAR120 covers the use of Marc and Patran or AFEA (the interlocked combination of Patran and Marc) for the solution of complex engineering problems. Students who successfully complete this course will be able to: create finite element models representing nonlinear physical phenomena; select appropriate element types and mesh densities; understand the limitations of solving nonlinear FEA problems; select solution types for various nonlinear phenomena such as nonlinear dynamics, metal forming, elastomers, and contact problems; select error tolerance parameters and properly use automatic time-stepping techniques; and understand the basis of large deformation, rotation, and strain finite element analysis.

Patran provides a Marc Preference which directly supports most Marc features and indirectly supports all Marc features. MSC customers that have been using Advanced FEA (which is replaced by AFEA) for meeting their analysis needs will find this new Marc Preference to be the ideal environment to continue their work.

They are especially encouraged to attend this course. All the class practice (16 exercises) is made using Patran and Marc rather than Marc and Mentat. Engineers who have attended the MAR101 and MAR102 will also benefit from attending this class if they intend to use the Patran Marc Preference.

Content

- One-day Patran overview
- Element formulation: selection and usage inside Patran and Marc
- Multistepping: how to setup and run prescribed loading history analysis
- Contact: in depth coverage of setup and solution of various contact types including rigid and elastic body contact, small and finite sliding, and 1, 2, and 3 dimensional contact modes
- Fundamentals of non-linear analysis, problem formulation and convergence

- Introduction into material modeling capabilities including: elastic, plastic, hyperelastic, creep, and composite material models
- Introduction to linear and non-linear direct dynamics and modal dynamic analysis methods
- Basic introduction to steady state and transient heat transfer analysis

Prerequisites Basic knowledge of statics and strength of materials is highly recommended and previous finite element analysis experience is recommended.

Duration 4 Days

Price $1,900

View Schedule

- Global Remeshing
- Local Adaptive Remeshing
- Creep of a Tube (User Subroutine)
- Heat Transfer (Conduction and Convection)
- Coupled Analysis (Thermal / Structural)
- Global – Local (Structural Zooming)
- Restarts

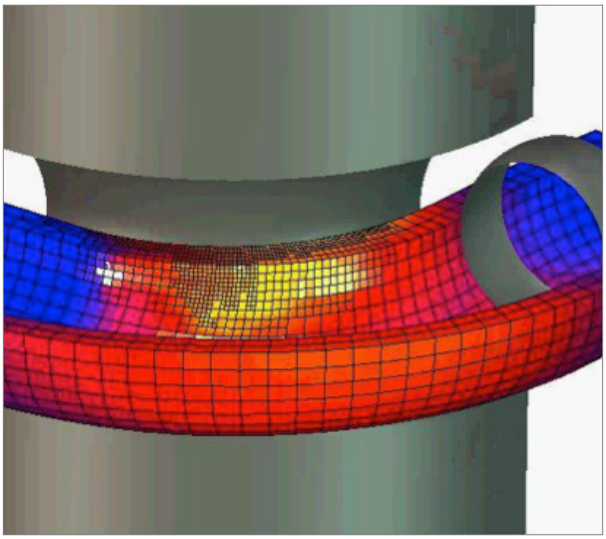
Prerequisites MAR120 - Basic Nonlinear Analysis using Marc and Patran.

Duration 3 Days

Price $1,425

View Schedule

MAR121 - Advanced Nonlinear Analysis using Marc and Patran



Objective

The purpose of this course is to enhance the current Marc user's understanding of modeling nonlinear problems. Lectures are supported by hands-on modeling of nonlinear problems.

Content

- Expand knowledge from MAR120 (Basic Nonlinear Analysis using Marc and Patran) course
- Advanced subjects including:
  - Practical aspects of rubber simulation
  - Creep
  - Superplastic forming
  - Composite failure techniques
  - Advanced contact techniques
  - Adaptive meshing
  - User subroutines
  - Global/Local modeling
  - Heat transfer and thermal stress
  - Coupled Thermal/Structural analysis
  - Restarts
  - Performance
- Workshop Problems
  - Experimental Curve Fitting Using Physical Test Data
  - Creep of Tube
  - Superplastic Forming of a Metal Container
  - Composite Progressive Failure Analysis using VCCT
  - Elastomeric Cylinder (Segment to Segment Contact)
  - Ship Bumper Contact Analysis

MaterialCenter

MAT101 - Introduction to MaterialCenter

MaterialCenter is a process and data management system for material data. This course covers the Web User Interface of MaterialCenter to navigate, search, plot, analyze, and retrieve data to assist engineers in their material down-selection process.

Content

- Introduction and Overview of MaterialCenter
- Web User Interface and Home Page
- Content Management in MaterialCenter
- Clipboards and Folders
- Search and Find Similar
- Plotting and Curve Viewing
- Material Data Comparison

Prerequisites None

Duration 1 Day

Price $475

View Schedule

MAT102 – MaterialCenter for the Materials Engineer

Objective

This students on this course will be able to use the Web User Interface and integrations of MaterialCenter to create material schemas, publish and retrieve test and design data, generate CAE material cards. They also will be able to create and manage approval workflows and projects as well as define user profiles.

Content

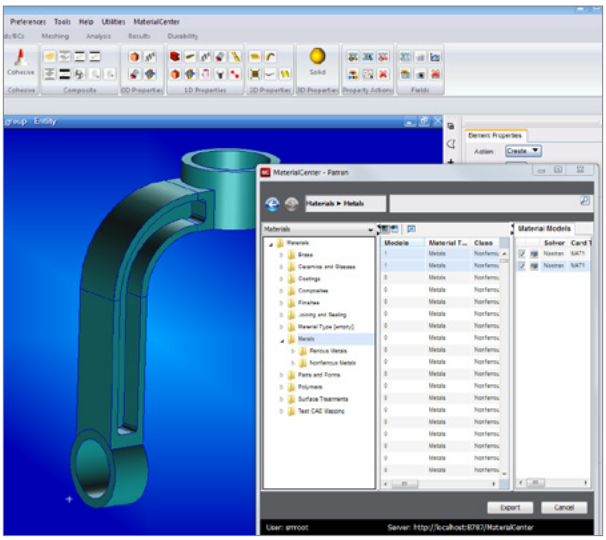
- Create Schema and Material Records
- Excel Integration
- Test Data Management
- Create and Export CAE Data using Approval Tools
- Create, Assign, and Track Work Requests
- Mvision Databank Support

Prerequisites None

Duration 1 Day

Price $475

View Schedule



MSC Nastran

NAS101A - Linear Static and Normal Modes Analysis using MSC Nastran

Objective

This course serves as an introduction to finite element analysis. It includes discussion of basic features available in MSC Nastran for solving structural engineering problems. In this course, all finite element models will be created and edited using a text editor, not a graphical pre-processor. Proper data structure of the MSC Nastran input file is covered. At the conclusion of seminar, the student will be familiar with fundamental usage of MSC Nastran.

Content

- Introduction to MSC Nastran
- Finite Element Overview
- Introduction to the File Management, Executive, Case Control, and Bulk Data sections of the input file
- Model Verification
- Normal Modes Analysis

Prerequisites A basic knowledge of statics and strength of materials is highly recommended. No previous finite element analysis experience is required.

Duration 3 Days

Price $1,425

View Schedule

NAS101B - Advanced Linear Analysis using MSC Nastran

Objective

This course is a continuation of NAS101A - Linear Static and Normal Modes Analysis using MSC Nastran. In this class, you will learn:

- Theory of buckling analysis and how to perform a buckling analysis
- About rigid elements – MPC, RBAR,RBE2, and RBE3
- Modeling with interface element CINTC and connectors
- Lamination theory and composite materials
- MSC Nastran composite theory
- Failure theories
- Linear contact and permanent glued contact
- Different model checks
- Modeling tips and tricks

Content

- Linear buckling analysis
- Understanding MPCs, Connectors and R-Elements
- Composite Analysis
- Linear Contact and Permanent Glue
- Model Checkout and Tricks
  - Common Types of Errors
  - Stiffness Matrix Checks
  - How to Avoid Serious Modeling Mistakes
  - Check for Bad Modes
  - Displacement Coordinates Systems
  - Offsets on Bars, Beams, and Plates
  - Plate to Solid Transition

Prerequisites NAS101A (Linear Static and Normal Modes Analysis using MSC Nastran) or equivalent.

Duration 1 Day

Price $475

View Schedule

NAS102A - Dynamic Analysis using MSC Nastran

Objective

In this class, you will be exposed to various aspects of dynamic analysis using MSC Nastran. This includes normal modes, frequency response, transient response, and enforced motion.

Content

- Review of Fundamentals
- Dynamic Modeling Input
- Normal Mode Analysis
- Normal Modes of Preloaded Structures
- Reduction in Dynamic Analysis
- Rigid Body Modes
- Damping
- Transient Response Analysis

- Frequency Response Analysis
- Dynamic Equations of Motion
- Residual Vector Methods
- Enforced Motion
- Shock and Response Spectrum Analysis
- Random Response Analysis

Prerequisites NAS101A (Linear Static and Normal Modes Analysis using MSC Nastran) and NAS101B (Advanced Linear Analysis using MSC Nastran), or equivalent.

Duration 3 Days

Price $1,425

View Schedule

NAS102B - Advanced Dynamic Analysis using MSC Nastran

Objective

In this class, you will be exposed to various aspects of dynamic analysis using MSC Nastran. This includes frequency response, direct matrix input, modal effective mass, complex eigenvalue analysis, dynamic optimization, and test analysis correlation.

Content

- Review of Basic Dynamic Topics
  - Mass Properties
  - Damping
  - Transient Response
  - Frequency Response
- Direct Matrix Input
- General Dynamic Tools
  - Modal Effective Mass
  - Mass Check
  - Mode Selection
  - Grid Point Kinetic Energy
  - Modal Participation Factor
  - Using strain energy to identify dominant modes
  - Other model check out tools
- Residual Vectors
- Complex Eigenvalue Analysis
- Extra Points, Transfer Functions, and NOLINs
- Test-Analysis Correlation
- FRF/FBA/TPA
- Introduction to Nonlinear Dynamics
- Special Topics in Dynamics
  - Superelements
  - Rotordynamics
  - Acoustics
  - Design Optimization
  - Monitor Points

Prerequisites NAS101A (Linear Static and Normal Modes Analysis using MSC Nastran), NAS101B (Advanced Linear Analysis using MSC Nastran), and NAS102A (Dynamic Analysis using MSC Nastran); or equivalent.

Duration 2 Days

Price $950

View Schedule

NAS103 - MSC Nastran Nonlinear Analysis

Objective

This seminar provides a working knowledge of the nonlinear capabilities of MSC Nastran for static and dynamic analysis. Both geometric and material nonlinearity are discussed in detail. Nonlinear features of MSC Nastran elements are explained and several examples are presented. Some practical guidelines for nonlinear analysis are also given.

Content

- Day One
  - Introduction
  - Nonlinear static analysis strategies
    - Newton-Raphson method
    - Advancing schemes
    - Stiffness update schemes
    - Line search
    - Convergence and divergence
    - Restarts
- Day Two
  - Geometric nonlinear analysis
    - Large rotations
    - Follower forces
  - Linear and nonlinear buckling analysis
- Day Three
  - Material types
    - Nonlinear elastic
    - Hyperelastic (Green elastic)
    - Elastoplastic
    - Creep
    - Temperature dependent

- Nonlinear elements
  - Small strain elements
  - Large strain elements
  - Contact (interface) elements
  - Gap
  - Slideline
- Day Four
  - Nonlinear transient analysis
    - Integration schemes
    - Mass, damping, and load specification
    - Restarts
  - Superelements
  - Special topics
    - Nonlinear modal analysis
    - Composite elements

Prerequisites A working knowledge of linear static analysis as covered in NAS101. Experience with dynamic analysis and superelement analysis is recommended.

Duration 4 Days

Price $1900

View Schedule

NAS104 - Thermal Analysis using MSC Nastran (SOLs 153 and 159)

Objective

This seminar describes the latest heat transfer and thermal stress analysis capabilities in MSC Nastran. Program inputs and interpretation of results for conduction, convection, and radiation analyses are covered in detail. The seminar attempts to provide a balance between theory, its development within the context of MSC Nastran, and practical application. Example problems are used to clarify the information presented.

Content

- Day One
  - Introduction to MSC Nastran
  - MSC Nastran communication
    - Input data formats
    - Executive control statements
    - Case control statements
    - Parameter statements
  - Overview of heat transfer capabilities
    - Conduction
    - Convection
    - Radiation
  - The MSC Nastran thermal model
    - Geometry grid points and elements
    - Material properties
    - Boundary conditions surface elements
    - Thermal “loads”
    - Thermal transients
- Day Two
  - NASTRAN input data
  - Steady state analysis (SOL 153)
    - NLPARM statement
    - Free and forced convection
- Day Three
- Steady state analysis (SOL 153) (cont.)
  - Thermal loads
  - Radiation boundary condition
  - Radiation view factors
  - Radiation enclosure analysis
  - Spectral exchange
- Transient analysis (SOL 159)
  - TSTEPNL statement
  - Transient load methodology
  - Control nodes
  - Temperature boundary conditions
  - Phase change
- Day Four
  - Thermal stress analysis
    - Restarts
- Multimode heat transfer analysis
- Thermal system analysis
- Miscellaneous topics

Prerequisites Basic knowledge of heat transfer fundamentals. NAS101A (Linear Static and Normal Modes Analysis using MSC Nastran) or equivalent experience is recommended.

Duration 4 Days

Price $1,900

View Schedule

NAS106A - Basic Substructure Analysis using MSC Nastran - Primary Superelements

Objective

In this class, you will learn how to define superelements, analyze superelements in static and dynamic analyses (including component modes), and set up both single and multi-level superelement analysis. You will learn how to perform restarts using superelements and incorporate superelements with nonlinear analysis.

Content

- Defining Superelements and Terminology
- Loads, Constraints, Parameters, and Case Control
- Multilevel Superelement Analysis
- Component Modes, Reduction, and Assembly
- Model Checkout Tools
- Database and Restarts
- Dynamic Analysis with Superelements
- Nonlinear Analysis

Prerequisites NAS101A (Linear Static and Normal Modes Analysis using MSC Nastran), NAS120 (Linear Static Analysis using MSC Nastran and Patran), or equivalent.

Duration 2 Days

Price $950

View Schedule

NAS106B - Advanced Substructure Analysis using MSC Nastran - Secondary Superelements

Objective

This class deals primarily with secondary superelements. At the end of the class, you will be able to:

- Create external superelements
- Assemble external superelements and analyze the combined structure along with selective data recovery.
- Run Static, Dynamic, and Temperature loading on External superelements
- Use External Component in Optimization Solution Sequence
- Create and Use Image and Mirror Superelements.

Content

- External Superelement
- Additional Topics and Special Cases for External Superelements
- Image Superelements

Prerequisites NAS101A (Linear Static and Normal Modes Analysis using MSC Nastran), NAS120 (Linear Static Analysis Using MSC Nastran and Patran), or equivalent; NAS106A (Basic Substructure Analysis using MSC Nastran - Primary Superelements.

Duration 1 Day

Price $475

View Schedule

NAS107 - Design Sensitivity and Optimization using MSC Nastran

Objective

MSC Nastran features a comprehensive design sensitivity and optimization capability. It is possible to design for a variety of user-defined objectives such as minimum weight or maximum frequency. This seminar covers the theoretical and practical aspects of MSC Nastran design sensitivity and optimization-emphasis is placed on using the program to solve practical engineering problems. The capabilities of the program, including typical applications, are covered in detail. The concept of a design model is introduced. The process of optimizing a structure is discussed from initial modeling to interpretation of results. Numerous example problems reinforce the material covered in the lectures.

Content

- Day One
  - Introduction
  - What is design optimization?
  - Overview of MSC Nastran design optimization capabilities
  - The basic optimization problem statement
  - Introduction to numerical optimization
  - The concept of a design space
  - Constrained and unconstrained minimization techniques
  - Convergence testing
  - Numerical aspects of interest to the design engineer
  - Design modeling I Analysis versus design modeling
  - Design model definition procedure—choosing the design variables, objective, and constraints
  - Design optimization input, part I
- Day Two

- Design modeling II
- Design variable and reduced basis formulations
- Formulation of synthetic design variable-to-property relations
- Formulation of synthetic responses
- Design optimization input, part II
- Cautions and techniques for avoiding common pitfalls
- Interpretation and utilization of optimization results
- Structural Optimization—theory and practice
- Coupling numerical optimizers and structural analysis—the theory of structural optimization
- Introduction to approximation concepts
- Design variable View Scheduling
- Constraint deletion
- Formal approximations
- Taking advantage of the approximation concepts in MSC Nastran
- Design sensitivity analysis

• Day Three
- Shape optimization
- Introduction and theory
- Shape basis vectors in the design model
- Overview of methods
- Dynamic response
- Optimization
- General considerations
- Basic equations: direct frequency, modal frequency, and modal transient response optimization
- Superelement optimization
- Overview of capabilities and general considerations
- Case control structure

Prerequisites NAS101A (Linear Static and Normal Modes Analysis using MSC Nastran) or equivalent experience.

Duration 3 Days

Price $1,425

View Schedule

NAS110 - Working with Custom MSC Nastran Solution Sequences using DMAP

Objective

The Direct Matrix Abstraction Program (DMAP) is a macro language used to build and modify solution sequences in MSC Nastran. The DMAP language contains powerful matrix instructions as well as flexible scalar variable (parameter) operations. The File Management Section (FMS) is used for the attachment, initialization, and manipulation of databases.

The objective of this seminar is to present DMAP and database application techniques and to prepare attendees to develop DMAP and solution sequence alter packages. This seminar also provides experienced users with the knowledge to perform sophisticated tasks in MSC Nastran. Such tasks include the creation of DMAP sequences with subDMAPs and the creation of a solution sequence (delivery) database. Details are presented on the structured solution sequence (SOLs 100 through 200), DMAP structure, and NDDL.

Content

• Day One
- Introduction
  - Sample DMAP instructions and sequence
  - FORTRAN versus DMAP
  - What's new for DMAP and FMS

• Fundamentals
- Module and statement format and syntax
- Module property list (MPL)
- Parameters
- Assignment, expressions, operators, and functions
- Control statements
- Data block type and status

• Utility modules
- APPEND module and FILE statement
- EQUIVX and COPY modules
- VECPLOT module
- DELETE and PURGEX modules
- PARAML module
- MODTRL module modify trailer
- MATGEN and MATMOD modules

• Day Two
- Matrix modules
- Input/output to a DMAP sequence
- Examples of a DMAP Sequence
- Parameter specification

• Day Three
- Looping in a DMAP sequence
- According to case control
- According to superelement
- Looping with PARAML module

• How to CALL a SubDMAP
- How to compile and View Schedule with called SubDMAP
- Qualifier value setting and scope

- DBLOCATE and ACQUIRE FMS statements
- How to store a data block
  - Brief description of the NDDL
  - NDDL statements and simplified NDDL example
  - The DBMGR SubDMAPs
- Structured solution sequences
- Common user errors and pitfalls
- Diagnostic tools and debugging
- Day Three (Optional)
- Advanced topics selected from the following (to be determined by instructor and attendees)
  - How to create a user delivery
  - What is an empty data block?
  - Data block structure
  - DEPEN statement and anatomy of a DEPENdency
  - LOCATION parameters and DBSET specification
- Appendices

Prerequisites NAS101A (Linear Static and Normal Modes Analysis using MSC Nastran) and NAS101B (Advanced Linear Analysis using MSC Nastran) or equivalent experience. Experience with additional solution sequences and super-element analysis is helpful.

Duration 3 Days

Price $1,425

View Schedule

NAS111 - Aeroelasticity using MSC Nastran

Objective

This seminar is intended for engineers concerned with structural loads, flying qualities, and aeroelastic stability of flexible aircraft and missiles. The objective of the seminar is to familiarize the engineer with state-of-the-art MSC Nastran applications in aeroelastic analyses.

An overview of the aeroelastic capability is followed by an in-depth discussion of the available aerodynamic theories and the three available aeroelastic solutions: static aeroelasticity, flutter, and dynamic aeroelasticity. Advanced topics include modeling aerodynamic bodies, active control systems, and the specification of dynamic loads, sensitivity analysis, and aeroelastic optimization.

Content

- Day One
  - Overview of MSC Nastran aeroelastic capabilities
    - Historical development
    - Solution sequences
    - Set definitions
    - Matrix notation
    - Coordinate systems
  - Aerodynamic theories
    - Subsonic doublet-lattice method (DLM)
    - DLM with body interference
    - ZONA51
    - Strip theory
    - Mach box method
    - Piston theory
  - Surface and linear splines
  - Static aeroelastic analysis
    - Theoretical background
    - Trim
    - Stability derivatives
    - Element loads and stresses
    - Divergence analysis
    - Preparation of input/sample problems
    - Stiffness and mass data
    - Aerodynamic data
    - Extra points for control surfaces
    - Spline data
    - Direct matrix input
    - Solution control
- Day Two
  - Flutter analysis
    - Methods of flutter analysis
    - Structural damping
    - Estimation of dynamic stability derivatives
    - Inclusion of flight control systems
    - Transfer functions
  - Preparation of input/sample problems
    - Calculation of modes
    - Complex eigenvalues
    - Aeroelastic divergence
    - Solution control
  - Guidelines for flutter analysis
    - Selection of flutter method
    - Convergence of modal solution
    - Aerodynamic modeling
- Day Three
  - Dynamic aeroelasticity
  - Preparation of input/sample problems

    - Dynamic loads data
    - Transient response calculation
    - Gust response calculation
  - Guidelines for effective response analysis
    - Periodic loading
    - Frequency distribution
    - Spectral inputs
  - Aeroelastic design sensitivities and optimization
  - Miscellaneous topics

Prerequisites NAS101A (Linear Static and Normal Modes Analysis using MSC Nastran), NAS101B (Advanced Linear Analysis using MSC Nastran), and NAS102A (Dynamic Analysis using MSC Nastran).

Duration 3 Days

Price $1,425

View Schedule

NAS113 - Composite Material Analysis with MSC Nastran

Objective

MSC Nastran can be efficiently and effectively used to model laminated composites. MSC Nastran allows the user to specify the material properties, orientation, and thickness for each lamina in the composite layup. The program then calculates the properties of the equivalent plate. This automatic representation of laminated composites is available in all MSC Nastran solution sequences; however stresses, strains, and failure indices can be recovered at the lamina level for only statics, normal modes, and nonlinear statics.

In addition, MSC Nastran can be used for multi-disciplinary structural optimization of laminated composite materials. This seminar describes how to use MSC Nastran for practical analysis and design optimization of composite materials. Examples are provided that illustrate typical uses for all major topics. The Patran composite pre- and post processing is also shown.

Content

- Introduction to composites in MSC Nastran
- Composite bulk data entries
- Overview of classical lamination theory
  - Industry definitions
  - Application to MSC Nastran
- Composite post-processing
  - Lamina stresses and strains
  - Interlaminar shear stresses and strains
- Ply failure theories
  - Hill, Hoffman, Tsai-Wu, and maximum strain
- Interlaminar shear
- Modal analysis with composites
- Nonlinear composite analysis
- Optimization of composites
- Composite use in other solution sequences
- Laminate Modeler

Prerequisites NAS101A (Linear Static and Normal Modes Analysis using MSC Nastran) or equivalent experience.

Duration 3 Days

Price $1,425

View Schedule

NAS115 - Fluid Structure Analysis using MSC Nastran

Objective

Acoustics is the study of noise. In this class you will be exposed to various aspects of acoustics including the coupling and interaction of acoustics with structures.

Content

- Interior Acoustics
- Exterior Acoustics
- Coupling of structure to fluid
- Modal and Direct Approach
- Participation Factors
- Equivalent Radiated Power
- Virtual Mass
- High Frequency Acoustics
- Acoustic Optimization

Prerequisites NAS101A (Linear Static and Normal Modes Analysis using MSC Nastran),
NAS101B (Advanced Linear Analysis using MSC Nastran),
NAS120 (Linear Static Analysis Using MSC Nastran and Patran), or equivalent experience.

Duration 3 Days

Price $1,425

View Schedule

NAS120 - Linear Static Analysis using MSC Nastran and Patran

Objective

This seminar introduces basic finite element analysis techniques for linear static, normal modes, and buckling analysis of structures using MSC Nastran and Patran. MSC Nastran data structure, the element library, modeling practices, model validation, and guidelines for efficient solutions are discussed and illustrated with examples and workshops. Patran will be an integral part of the examples and workshops and will be used to generate and verify illustrative MSC Nastran models, manage analysis submission requests, and visualize results. This seminar provides the foundation required for intermediate and advanced MSC Nastran applications.

Content

- Introduction to finite element theory
- Introduction to finite element modeling
- Anatomy of the MSC Nastran input file
  - MSC Nastran statements
  - File management section
  - Executive control section
  - Case control section
  - Bulk data section
- Model generation and verification using Patran
  - Patran overview
  - Defining the analysis reference system: global versus local
  - Creating geometric representation of structure
  - Defining material and element properties
  - Constraining the model
  - Loading the model
  - Meshing the model
  - Model verification
  - Generating and submitting a ready-to-run MSC Nastran input file
- Linear static analysis
  - Introduction to linear static analysis
  - Input entries required for linear static analysis
- Normal modes analysis
  - Introduction to real eigenvalue analysis
  - Input entries required for real eigenvalue analysis
- Buckling analysis
  - Introduction to linear buckling analysis
  - Input entries required for linear buckling analysis
- Importing an existing MSC Nastran input file into Patran
- Anatomy of the .f06 file: MSC Nastran output interpretation
- Results visualization using Patran
  - Managing results visualization using viewports
  - Animated deformation and stress contour plots
  - Using visualization tools for model validation
  - User-derived results from MSC Nastran results
  - X-Y plots of user-selected results
- Model debugging tools and recommendations
- Tips for solving large problems

Prerequisites No previous finite element analysis or computer-based modeling experience is required. A basic knowledge of statics and strength of materials is highly recommended.

Duration 5 Days

Price $2,375

View Schedule

NAS122 - Basic Dynamic Analysis using MSC Nastran and Patran

Objective

The course covers a wide range of dynamic analysis topics from basic to advanced using an integrated approach. Patran is used for data set up and post-processing and MSC Nastran is used for the solver. Many unique practical hints and tips are given which do not exist in other material. Case studies are used in each topic to help understand the physics and engineering behind the techniques in a practical way. A comprehensive set of over 20 fully detailed student workshops is used to obtain real “hands on” experience. A strong emphasis is placed on engineering process so that the student can rapidly relate the course to his or her project needs.

Content

- Review of Fundamentals
- Normal Modes Analysis
- Mass Modeling
- Effective Mass
- Guyan Reduction
- Rigid Body Modes
  - Rigid Body Modes and Rigid Body Vectors
  - Calculation of Rigid Body Modes
  - Selection of “Support” Degrees of Freedom
  - Checking of “Support” Degrees of Freedom
  - Rigid Body Modes
- Pre-Stiffened Normal Modes
- Response Methods

- Transient Analysis
  - Frequency Response Analysis
  - Response Types
- Modal and Direct Methods
- Damping Overview
  - Damping in Dynamic Analysis
  - Rayleigh Damping
  - Viscous Damping Input
  - Frequency Dependent Impedance Sample
  - Sample using CBUSH Element
- Transient Response Analysis
- Frequency Response Analysis
- Enforced Motion
- Interactive Frequency Response
- Random Analysis

Prerequisites Some familiarity with Patran and MSC Nastran

Duration 5 Days

Price $2,375

View Schedule

NAS123 - Implicit Nonlinear Analysis using MSC Nastran (SOL600)

Objective

This seminar presents how to perform implicit nonlinear analysis using MSC Nastran SOL600. MSC Nastran SOL600 is the nonlinear capabilities of Marc delivered in an MSC Nastran user interface. SOL600 provides FEA capability for the analysis of 3D contact and highly nonlinear problems.

Content

- Overview of Nonlinear Analysis Using MSC Nastran SOL600
- Numerical Concepts in Nonlinear Analysis
- Element Library, Mesh Considerations, and Analysis Procedures
- Analysis Setup and the Analysis Form
- Analysis Setup for Non-Patran Users
- Introduction to Choice of Elements
- Choice of Element Integration
- Materials
- Overview and Contact Body Interactions
- Contact Body Definition
- Resolving Convergence Problems
- Buckling and Post-buckling Analysis
- Structural Dynamics

Prerequisites NAS101A (Linear Static and Normal Modes Analysis using MSC Nastran), NAS101B (Advanced Linear Analysis using MSC Nastran) or equivalent is recommended.

Duration 3 Days

Price $1,425

View Schedule

NAS124 - Thermal Analysis using MSC Nastran (SOL400)

Objective

This 3 day seminar describes heat transfer and thermal stress analysis capabilities in MSC Nastran's SOL400. Program inputs and interpretation of results for conduction, convection, and radiation analyses are covered in detail.

The seminar attempts to provide a balance between theory, its development within the context of MSC Nastran, and practical application. Example problems are used to clarify the information presented.

Content

- Introduction to Thermal Analysis
- Surface Elements, Constraints, and Loading
- Conduction with Elements and Materials
- Convection Thermal Analysis
- Transient Thermal Analysis
- Radiation Thermal Analysis
- SOL400 Coupled Load Cases
- Thermal Analysis Theory

Prerequisites Basic knowledge of heat transfer fundamentals. NAS101A (Linear Static and Normal Modes Analysis using MSC Nastran) or equivalent experience is recommended.

Duration 3 Days

Price $1,425

View Schedule

NAS126 - Explicit Nonlinear Analysis (SOL700) using MSC Nastran and Patran

Objective

NAS126 is an introductory course in dynamics simulation using Explicit nonlinear analysis. Students will prepare short duration structural dynamic analyses using MSC Nastran finite element models. Students will learn how to create and/or modify material properties, loads, and boundary and initial conditions for dynamic simulation models. They will also learn how to set up the jobs for running the models and review the results for these models.

Content

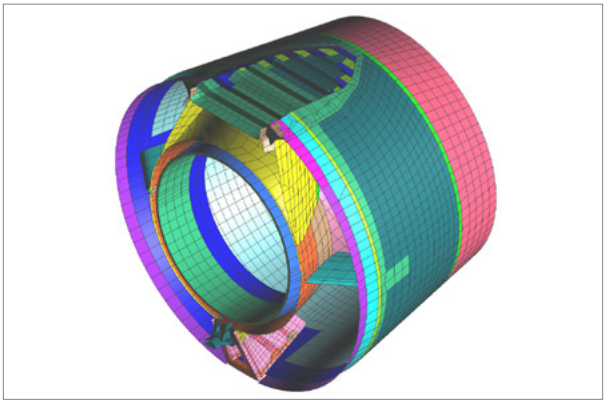
- Explicit Transient Dynamic Analysis
- Input file format
- Patran Graphical User Interface
- Running MSC Nastran SOL700
- Lagrange Basics
- Element Library
- Material Models
- Lagrangian Loading Conditions
- Lagrangian Boundary Conditions
- Contact Definition and Analysis

Prerequisites NAS120 (Linear Static Analysis Using MSC Nastran and Patran), or PAT301 (Introduction to Patran), NAS101A (Linear Static and Normal Modes Analysis using MSC Nastran), and NAS101B (Advanced Linear Analysis using MSC Nastran), or equivalent.

Duration 3 Days

Price $1,425

View Schedule



NAS127 - Rotordynamic Analysis using MSC Nastran

Objective

- This course covers:
  - Rotordynamic analysis for coupled rotating and stationary components
    - For example: Jet Engines, Turbines, Compressors, Energy Storage Devices
- How to setup and analyze structural models with one or more rotating components
- The types of analysis supported by the rotordynamics capability
  - Static analysis
  - Complex eigenvalue analysis (modal and direct)
  - Frequency response (modal and direct) and nonlinear frequency response
  - Transient response (direct linear and nonlinear)
- Damping effects and input methods for models with rotating components
- How to use Patran to create models and display results with animation or graphs
- There are two sets of workshops for this course. One set uses Patran as the pre- and post-processor. The other directly edits the MSC Nastran input file.

Content

- Theory
  - Static Analysis
  - Complex Eigenvalue Analysis
  - Frequency Domain Analysis
  - Time Domain
  - Multiple and Reference Rotors
- New Forms of Damping Input
  - Damping Input Entries
  - Rotordynamic Bulk Data Entries
  - Parameters
- Patran Support
- Example Cases
  - Line Model without Superelements
  - 3D Example
  - Frequency Response
  - 3D-2 Rotor Model
  - Typical Commercial Jet Engine
  - Hybrid Damping
- Nonlinear Frequency Response

Prerequisites NAS120 (Linear Static Analysis Using MSC Nastran and Patran) and NAS122 (Basic Dynamic Analysis Using MSC Nastran and Patran) or NAS102A (Dynamic Analysis using MSC Nastran) and NAS102B (Advanced Dynamic Analysis Using MSC Nastran).

Duration 2 Days

Price $950

View Schedule

NAS133 - Contact Analysis using MSC Nastran and Patran

Objective

In this class you will be exposed to different ways of using contact in MSC Nastran's Linear Static and Normal Modes solutions. This includes setting up contact bodies, both touching and glued contact, and interpretation of results. Patran is used for pre- and post-processing.

Content

- The above concepts are reinforced with various case studies and workshops.
  - Topics:
    - Concepts of Contact in Linear MSC Nastran Analysis
    - Defining and Creating Contact Bodies
    - Job Setup
    - Contact Detection
    - Contact Tables
    - Fundamental Contact Controls to Obtain Convergence in Linear Analysis
    - Glued Contact
    - Shell Contact
    - Friction
    - Interference Fit
    - Contact Results Output

Prerequisites NAS120 (Linear Static Analysis Using MSC Nastran and Patran) or PAT301 (Introduction to Patran), NAS101A (Linear Static and Normal Modes Analysis using MSC Nastran), and NAS101B (Advanced Linear Analysis using MSC Nastran) or equivalent.

Duration 2 Days

Price $950

View Schedule

NAS134 - Advanced Contact Analysis using MSC Nastran and Patran (With Contact Tables or Contact Pairs)

Objective

In this class, you will be exposed to different ways of using contact in MSC Nastran and Patran. This includes setting up contact bodies, both touching and glued contact, and interpretation of results.

Content

- Contact Analysis
- Defining and Creating Contact Bodies
- Job Setup
- Discussion of Contact Detection
- Describing the Importance of Various Contact Search Options and Strategies
- Defining Touching and Glued Contact
- Discussion of Glued Contact Status
- Identifying Glued Contact Grounding and Modeling Issues
- Specifying Contact Separation Control

Prerequisites NAS101A - Linear Static and Normal Modes Analysis using MSC Nastran NAS101B - Advanced Linear Analysis using MSC Nastran PAT301 - Introduction to Patran NAS133 - Contact Analysis using MSC Nastran and Patran Or NAS120 - Linear Static Analysis using MSC Nastran and Patran NAS133 - Contact Analysis using MSC Nastran and Patran.

Duration 1 Day

Price $475

View Schedule

NAS318 - Implementation of Fatigue Method using MSC Nastran-Embedded Fatigue (NEF) with Patran

Objective

- Learn the proper use of MSC Nastran for solving various fatigue analysis problems
- Understand the physical processes in fatigue
- Use MSC Nastran to model the fatigue processes
- Become familiar with common fatigue methods

Content

- Vibration Environments - Random v Deterministic
- Frequency Time Domain

- System Transfer Functions (Frequency Domain)
- How Transfer Functions Work
- FFTs and PSDs
- Calculating PSDs From Time Signals - Buffers and Window Averaging
- Classifying Time Signals and PSDs
- Statistics, Probability, and Reliability
- Loads and “Cross PSDs”
- Gaussian, Random, and Stationary Data Zero and Peak Crossing Rates, Irregularity Factor, Root Mean Square (RMS) Value & Moments
- Hand Calculations From a PSD
- Options For Calculating Fatigue Life From PSDs
- Vibration Fatigue Solvers - Narrow Band, Wirsching, Steinberg, Dirlik
- Case Studies Related to Frequency Based Fatigue

Prerequisites NAS318 is an introductory class for MSC Nastran Embedded Fatigue

Duration 1 Day

Price $475

View Schedule

NAS319A - Fatigue & CAE Integration

A state of the art view of CAE based fatigue, durability and damage tolerance calculations will be presented with a particular emphasis on conceptual behavior. This class will show that the basic principles of fatigue life estimation are relatively straightforward and certainly within the role of typical stress, dynamics, and FEA design engineers. Although MSC products will be used to show example problems the course material will be independent of any software products and generally focused on the technology rather than specific software products.

Content

- History of Fatigue Methods and Their Integration into the CAE Environment
- The concept of FE Based Fatigue Analysis
- Background to Dynamics For Fatigue Analysis
- Step By Step Process For Static and Dynamic Models (Both Time and Frequency Domains)
- Background Fatigue (Stress Based)
- Materials Considerations
- Loading Considerations
- Stress-Life Based Fatigue in More Detail
- Strain-Life Based Fatigue
- Fracture Mechanics and Crack Propagation (Brief Treatment Only)
- Accelerated Testing Techniques
- Composites and Non-Metallic Materials
- An Introduction to Weld Analysis, Wheels Fatigue, Multiaxial Fatigue and MBS
- Offshore Structures and Fatigue

Prerequisites The course will give a basic overview of the methods – but from a quite unconventional perspective (concept rather than equation driven). So no prior knowledge of fatigue analysis techniques or dynamics is required. The latest fundamental developments (theories and implementations) in the field will also be presented.

Duration 1 Day

Price $475

View Schedule

NAS319B - Fatigue & Dynamics - The Reliability of Vibrating Systems

A state of the art view of fatigue calculations for dynamic systems and vibration environments (such as that defined in military standard MIL-STD-810). This course covers vibration fatigue which brings together structural dynamics, FEA, and fatigue analysis in order to show how systems with dynamic (resonant) response can be assessed for fatigue life. Although MSC products will be used to show example problems the course material will be independent of any software products and generally focused on the technology rather than specific software products.

Content

- Vibration Environments - Random v Deterministic
- Frequency v Time Domain
- System Transfer Functions (Frequency Domain)
- How Transfer Functions Work
- FFTs and PSDs
- Calculating PSDs From Time Signals - Buffers and Window Averaging
- Classifying Time Signals and PSDs
- Statistics, Probability, and Reliability
- Loads and “Cross PSDs”
- Gaussian, Random, and Stationary Data
- Zero and Peak Crossing Rates, Irregularity Factor, Root Mean Square (RMS) Value & Moments
- Hand Calculations From a PSD
- Options For Calculating Fatigue Life From PSDs

- Vibration Fatigue Solvers - Narrow Band, Wirsching, Steinberg, Dirlik
- Case Studies Related to Frequency Based Fatigue

Prerequisites The course will give a basic overview of the methods – but from a quite unconventional perspective (concept rather than equation driven). So no prior knowledge of fatigue analysis techniques or dynamics is required. The latest fundamental developments (theories and implementations) in the field will also be presented.

Duration 1 Day

Price $475

View Schedule

NAS319C - Practical Implementation of Fatigue Methods with MSC Nastran, Patran and Other Applications

Objective

- Learn the proper use of MSC Nastran for solving various fatigue analysis problems
- Understand the physical processes in fatigue
- Use MSC Nastran to model the fatigue processes
- Become familiar with common fatigue methods

Content

- Vibration Environments - Random v Deterministic
- Frequency Time Domain
- System Transfer Functions (Frequency Domain)
- How Transfer Functions Work
- FFTs and PSDs
- Calculating PSDs From Time Signals - Buffers and Window Averaging
- Classifying Time Signals and PSDs
- Statistics, Probability, and Reliability
- Loads and “Cross PSDs”
- Gaussian, Random, and Stationary Data Zero and Peak Crossing Rates, Irregularity Factor, Root Mean Square (RMS) Value & Moments
- Hand Calculations From a PSD
- Options For Calculating Fatigue Life From PSDs
- Vibration Fatigue Solvers - Narrow Band, Wirsching, Steinberg, Dirlik
- Case Studies Related to Frequency Based Fatigue

Prerequisites NAS318 is an introductory class for MSC Nastran Embedded Fatigue

Duration 1 Day

Price $475

View Schedule

NAS400 - Implicit Nonlinear Analysis using MSC Nastran and Patran

Objective

In this class, you will be exposed to various aspects of implicit nonlinear analysis using MSC Nastran and Patran. This includes large deformation, advanced nonlinear material, contact, analysis chaining, heat transfer, and nonlinear transient dynamics.

Content

- Day 1
  - Overview of MSC Nastran
  - Overview of Implicit Analysis with SOL 400
  - Sources of Nonlinearity
  - Concepts in Nonlinear Analysis
  - Chaining of Different Analysis Types
  - Boundary Condition Changes
- Day 2
  - Advanced Nonlinear Material Models in MSC Nastran
    - Large Strain Elastic-Plastic
    - Nonlinear Elastic
    - Hyperelastic
    - Composite
    - Creep
    - Shape Memory Alloy
  - Advanced Nonlinear Elements in MSC Nastran
- Day 3
  - Contact Bodies and Contact Body Interactions
  - Heat Transfer
  - Nonlinear Structural Dynamics

Prerequisites NAS120 (Linear Static Analysis Using MSC Nastran and Patran), or PAT301 (Introduction to Patran) and NAS101A (Linear Static and Normal Modes Analysis using MSC Nastran), or equivalent.

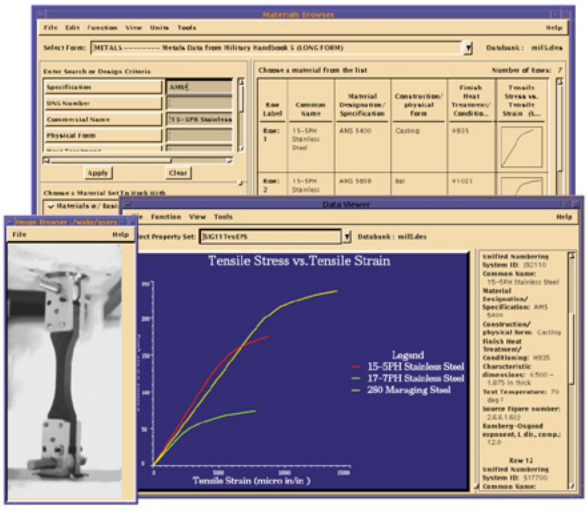
Duration 3 Days

Price $1,425

View Schedule

MVision

MVI320 – MVision Introduction



Objective

The purpose of this course is to provide an introduction to the use of the MVision materials system. This class is designed for engineers, scientists, and designers involved in the use of materials for analysis or design or those involved with the testing of materials.

Content

- Day One
  - Navigate MVision databanks
  - Query MVision databanks for materials meeting design constraints
  - Create custom tables from any information in the databanks
  - Introduction to the engineering spreadsheet
- Day Two
  - Understand the MVision database structure
  - Create new databanks including customized organization of the databank
  - Load an MVision database with input files or via the spreadsheet
  - Use the engineering spreadsheet
    - Evaluate materials options
    - Aid in databank construction
- Day Three
  - Create user-defined functions to customize the MVision Spreadsheet
  - Create spreadsheet templates
    - Automatically reduce test data
    - Add data to an MVision databank
- Manage units with MVision
- Create ISO compliant text output of your databank (STEP/ Express Translator)

Prerequisites None

Duration 3 Days

Price $1,425

View Schedule

Patran

PAT301 - Introduction to Patran

Objective

PAT301 is the introductory course for new Patran users. Students will master the basic skills required to use Patran in typical MCAE applications. PAT301 emphasizes practical skills development through comprehensive, hands-on laboratory sessions. Students will learn to build analysis models using Patran, define material properties, create boundary conditions, apply loads, and submit their job for analysis and post-process results.

Content

- Take a model through the complete design and analysis cycle
- Import CAD geometry and use it to create new geometry in Patran
- Learn graphics manipulation, imaging, and viewing techniques
- Practice using the different meshing techniques: isomesher, paver, and tetmesher
- Verify your finite element model
- Apply loads and boundary conditions
- Learn advanced techniques:
  - Apply distributed pressure loads
  - Vary thicknesses
  - Use groups and lists to filter and group entities
- Submit analysis
- View and manipulate results

Prerequisites A basic knowledge of statics and strength of materials is recommended.

Duration 5 Days

Price $2,375

View Schedule

PAT302 - Advanced Geometry, Meshing, Customization, and Variable LBCs using Patran

Objective

PAT302 provides an in-depth examination of the advanced features of Patran. Sample topics covered by PAT302 include: advanced Patran features usage for meshing and mesh refinement, use of various Patran Command Language (PCL) files for session customization, application of advanced geometric construction techniques, definition of fields to represent spatially varying functions for loads and boundary conditions, generation of constraint equations (MPCs) to define physical relationships, and creation of sophisticated multi-effect graphical images.

Content

- Learn user interface shortcuts and advanced capabilities that will increase your Patran efficiency and help you customize your work environment
  - Wildcards, keywords, list processor syntax, and command line options
  - Database/model manipulation using group transformation and database merge capabilities
  - Customizing the quickpick menu and start-up environment
- Discover advanced geometry modeling and CAD access features
  - Advanced construction and editing techniques help you repair non-topologically congruent (i.e., “dirty”) geometry
  - Learn the mathematical basis for the various geometry formats
  - Parametric cubic geometry and the Patran 2.5 convention
- Apply advanced finite element modeling capabilities
  - Mapped, automated and 2-1/2-D meshing, mesh smoothing
  - Global-local and thermal-structural modeling using FEM fields
  - Advanced node and element editing techniques
  - Multi-point constraints
- Post-processing and advanced graphical results display
  - Combine multiple animation images
  - Advanced capabilities of results, insight, XY plot, and text report
  - Learn exactly how the numerical results values are used to produce the graphical display, i.e., data averaging and extrapolation

- Patran Command Language (PCL) Basics
  - PCL programming in conjunction with Patran modeling
  - PCL syntax, directives, operators,variables,arrays and libraries
  - Write a PCL routine to automate parameterized geometry creations

Prerequisites PAT301 (Introduction to Patran) or equivalent experience in the use of Patran.

Duration 4 Days

Price $1900

View Schedule

PAT304 – Automating Tasks and Basic GUI Customization Using the Patran Programming Command Language (PCL)

Objective

PAT304 provides students with a comprehensive overview of the Patran Command Language (PCL) for site integration and user programming. Topics include basic PCL syntax, creation of user interface objects (widgets) such as forms, buttons, sidebars, etc., spawning remote processes from the Patran session, usage of database access calls, handling interrupts from an event-driven system, compiling, debugging, and code management. Students will build practical skills by performing 17 PCL programming exercises in multiple laboratory sessions. Exercises include the development of PCL code to create a fully parameterized finite element analysis model for use in shape optimization and design studies.

Content

- File I/O (create and parse a text file)
- Spawn UNIX command from Patran (i.e. start your own analysis code processing)
- Parametric Modeling (i.e. build same part but vary length, width or radius of the part)
- Dynamic memory allocation
- Create a Graphical User Interface (GUI)
- General PCL syntax and usage

Prerequisites PAT301 (Introduction to Patran) or comparable experience. Familiarity with C or FORTRAN or Pascal.

Duration 5 Days

Price $1900

View Schedule

PAT312 – Thermal Analysis Using Patran Thermal

Objective

Day 1 is an intensive review of Patran focusing on building heat transfer models for Thermal. Days 2 through 4 begin with an initial overview of Thermal capabilities followed by exposure to all features of Thermal accessed through Patran. Each lecture and lessons will instruct you how to setup, execute, and post process the results of a heat transfer analysis. Lessons increase in the level of detail and complexity through the week.

Content

- Analyzing models which include the four basic modes of heat transfer
  - Conduction
  - Convection
  - Radiation
  - Advection
- Exercising the two primary types of heat transfer analysis
  - Steady-state
  - Transient
- Defining thermal materials
  - Constant properties
  - Variable and nonlinear properties
  - Built in thermal material properties database
  - User defined material properties database
- Describing and applying available element types and options
  - 1D Conductive or advective
  - 2D Planar
  - 2D Axisymmetric
  - 3D Solid and shell
- Defining heat transfer loads and boundary condition that are either constant or variable for
  - Temperatures
  - Heat fluxes
  - Volumetric heat sources
  - Nodal sources
  - Convection heat transfer coefficients
  - Contact heat transfer coefficients
  - Advective flows
  - Thermally radiating surfaces
- Programming user supplied subroutines to
  - Customize a solution, output, or runtime computation
  - Access solution parameters and model properties during runtime

- Using the built in hydraulic network solver and its associated element and boundary definitions. Accessing and customizing control parameters
  - Define solution type
  - Define converge and performance criteria
  - Optimize viewfactor computation speed and size

Prerequisites Background using thermal analysis with either finite difference or finite element formulations. PAT301 (Introduction to Patran) or equivalent experience in the use of Patran.

Duration 5 Days

Price $2,375

View Schedule

PAT318A - Basic Durability and Fatigue Life Analysis Using MSC Fatigue

Objective

This course introduces methods for evaluation and estimation of fatigue life using MSC Fatigue. Various approaches for extending the useful life of a design are discussed. In addition, design optimization based on a uniform life concept, and selection and evaluation of material surface finish and treatments, will also be covered.

Content

- Overview of Durability and Fatigue Life Analysis
- Overview of Fatigue
- MSC Fatigue User Interface
- Stress-Life (S-N) Method
- Strain-Life (E-N) Method
- Vibration Fatigue
- Multiple S-N Mean Stress Curves
- Temperature Corrected Fatigue
- Aerospace Spectrum File Support
- Duty Cycle Analyzer
- Running in Batch Mode

Prerequisites Knowledge of engineering fundamentals, strength of materials, and machine design.

Duration 2 Days

Price $950

View Schedule

PAT318B - Advanced Durability and Fatigue Life Analysis Using MSC Fatigue

Content

- Fatigue Crack Propagation
- Multiaxial Fatigue
- Fatigue of Welded Structures
- Software Strain Gauge
- MSC Fatigue Utilities
- MSC Fatigue Wheels

Prerequisites PAT318A - Basic Durability and Fatigue Life Analysis Using MSC Fatigue or equivalent experience.

Duration 2 Days

Price $950

View Schedule

PAT325 - Composite Laminate Modeling using Patran

Objective

This seminar shows outlines of composites materials theory and the integration between FEM and composites materials. Illustrate the basic functions of Patran Laminate Modeler and Composite design in MSC Nastran. Engineers and material scientists involved in the design, analysis and manufacture of composite components and structures would benefit from this seminar.

Content

- Introduction to Laminate Modeler
- Introduction to Composite Materials
- Review of Material Constitutive Laws, and Laminate Stiffness
- Creation of Composite Model in Patran
- Exercise Session
- Failure Criteria for Composites
- Post-processing of Composite Analysis Results
- Exercise Session
- Creation of Ply Materials, and Layups with Laminate Modeler
- Exercise Session
- Plies on Doubly Curved Surfaces

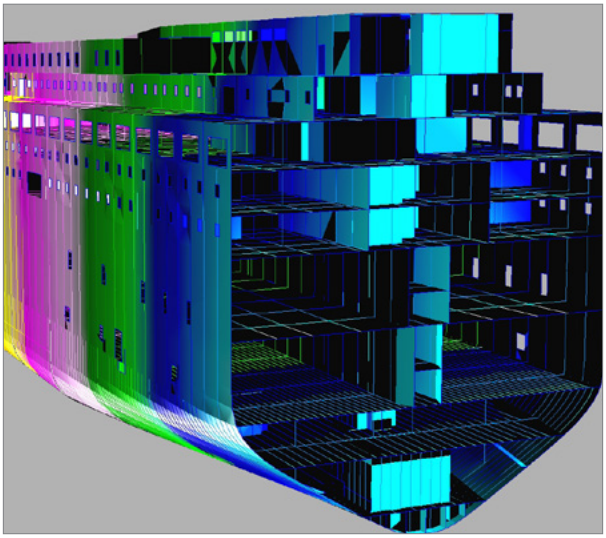
- Shear Algorithms
- Draping Algorithms
- Creating Splits in Plies
- Exercise Session
- Manufacturing Integration
- Exporting Flat Pattern
- Springback Analysis
- Exercise Session
- Optimization of Composites

Prerequisites A degree in engineering, material science or equivalent experience. Familiarity with Patran and a knowledge of composite materials.

Duration 2 Days

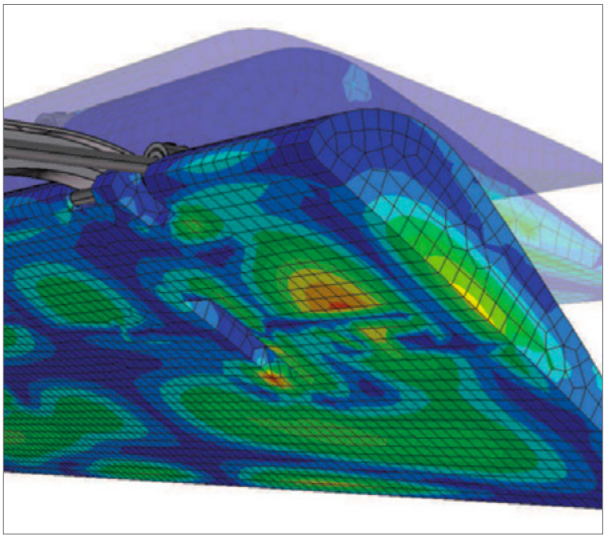
Price $950

View Schedule



SimDesigner

SMD102 - Fundamentals of Multibody Dynamics Analysis with SimDesigner



Objective

This course gives you the foundation you'll need to begin using SimDesigner Motion for CATIA V5. The backbone of the SimDesigner product is Adams; the most powerful virtual prototyping, testing, and visualization tool in the world.

Content

- Transform kinematics CATIA V5 models to Adams/Solver friendly mechanisms.
- Convert assembly constraints to mechanism constraints:
  - Standard joints (hinges, sliders, etc...)
  - Joint primitives
  - Complex (couplers, etc...)
  - Curve (cam-follower, pin-in-slot)
- Actuate a system with:
  - Ideal and complex part motion
  - Applied forces
  - Gravity
- Connect parts with more realistic forces:
  - Simple (springs, dampers, etc...)
  - Contacts and collisions
- Measure quantities of interest (displacements, velocities, accelerations, applied loads, forces)
- Precisely control and manage your simulations
- Investigate test results via animations and plots
- Manage files generated by exporting from CATIA V5 interface to Adams/View and/or Adams/Solver

Prerequisites This course is intended for users that have fundamental CATIA V5 knowledge. Experience with Part Design, Assembly Design and DMU Kinematics workbenches is recommended.

Duration 2 Days

Price $950

View Schedule

SimManager

SMM101 - Introduction to SimManager

Objective

By the end of this course, you will be able to:

- Use the Web User Interface of SimManager to navigate, search, and retrieve data
- Publish simulation data, such as models and results to SimManager
  - Using single publish actions
  - Using Import tools
- Collaborate and share data with other users
- Capture iterations and summarize results using the Study tool
- Understand the lifecycle stages of SimManager objects and the access controls used to manage security of data and actions
- Create Work Requests to define, assign, and track simulation tasks.

Content

- Introduction and Overview of SimManager
- Web User Interface and Home Page
- Content Management in SimManager
- Clipboard And Folders
- Importing and Exporting
- Project Hierarchy, Access Controls, And Lifecycle
- Study, Scenario, And Target
- Work Requests

Prerequisites None

Duration 1 Day

Price $475

View Schedule

SMM102 - SimManager Basic Configuration

Objective

By the end of this course, you will be able to use the web user interface of SimManager to perform the following administration and configuration tasks:

- Create and administer SimManager Projects to organize data and manage access
- Utilize Roles to define privileges for Users and Profiles
- Create and manage Users in SimManager
- Define groups of users with common privileges using Profiles
- Publish and maintain Procedures using the web UI
- Monitor user activity in SimManager
- Perform configuration tasks
- Manage Object types, enumerations, Variant naming, and queues.
- Publish configuration updates
- Integrate external applications
- Create and manage Work Request Templates.

Content

- Project administration
- User, Profile, and Role administration
- Monitors
- Procedure & Resource Management
- Configuration tasks
- Application Integration
- Work Request Templates

Prerequisites SMM101 (Introduction to SimManager)

Duration 1 Day

Price $475

View Schedule

SMM111 - Introduction to the SimManager Automotive Solution

Objective

This course covers the simulation automation capabilities of SimManager Automotive Solution. Participants will perform typical simulation and data management tasks performed by automotive analysts in SimManager Automotive Solution.

The “Import” action will be used to create a wide variety of SimManager objects and optionally perform a simulation. Students will then run the “Simulate” action consisting of the following steps: Solve, Post Process, and Reports. Complex actions are performed, such as “Import and Simulate”, in which the tasks of “Import” and “Simulate” are combined into one seamless operation.

Participants will utilize “Assemble and Simulate” to select a set of component models, then assemble them into a run-ready input deck, and perform a simulation. They will understand how to create and use a Product Structure Definition to define the templates for model assemblies and simulations. Students will utilize “Simulation Generator” to manage a complete set of simulations involving multiple load cases and possibly different model content and/or format.

Students will understand how SimManager Automotive Solution automatically stores simulation data and creates SimManager objects. They will learn how to traverse the object hierarchy and find simulation data. They will use the comparison and report tools to compare simulation data and automatically create comparison reports.

Content

- Import/export
- Set-up
- Simulate Overview
- Solve
- Post-processing
- Reports
- Import and Simulate
- Assemble and Simulate
- Compare tools
- Simulation Generator

Prerequisites SMM101 (Introduction to SimManager). Recommended: SMM102 (SimManager Basic Configuration).

Duration 2 Days

Price $950

View Schedule

SMM112 - SimManager Automotive Solution Configuration

Objective

This course covers the configuration of the SimManager Automotive Solution. Participants will set up a SimManager portal with a typical automotive product structure and enable simulation automation. The completed configuration is suitable for the related course: SMM111 (SimManager Automotive Solution).

Participants will configure a SimManager out-of-the-box installation for the Automotive Solution. They will set up automated simulation by integrating external applications, such as pre-processors, solvers, and post-processors. They will create some automotive-specific object types.

A master automobile project will be created and access controls will be established. Then a Product Structure Definition for an automotive vehicle will be created, which defines the typical components that comprise a vehicle assembly. Likewise, the necessary assembly and component models will be defined.

Students will define the scenarios (load cases) and expected key results from a typical crash simulation. They will create a Simulation Study, in which component models can be assembled and run through multiple simulations using the Simulation Generator. They will also enable server-side file selection (SSFS) for more efficient uploading of data.

Content

- Customer-specific object types
- Application Integration
- Automobile Program Master Project
- Product Structure Definition
- Scenario and Key Result Request Configuration
- Simulation Study Configuration
- Server Side File Selection Configuration

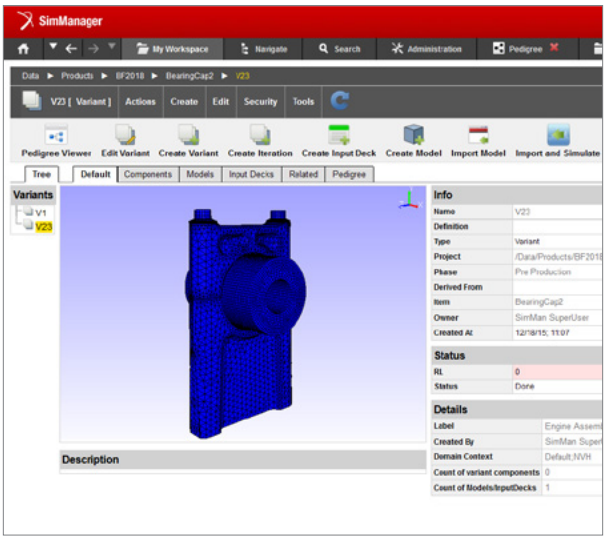
Prerequisites SMM101 (Introduction to SimManager), SMM102 (SimManager Basic Configuration). Recommended: SMM111 (Introduction to the SimManager Automotive Solution) - take either before, or immediately after SMM112.

Duration 1 Day

Price $475

View Schedule

SMM201 – SimManager Installation and Administration



Objective

This course is for installers and administrators of SimManager. Participants will learn the basic architecture and supported hardware environments. They will prepare for and install SimManager and test the installation. Studio will be used to administer and maintain the SimManager system.

They will learn how to create, delete, and migrate SimManager databases. Learners will be able to add, remove, move, and back-up data vaults. They will understand how to implement different user authentication mechanisms and to set up job queuing systems in SimManager. They will become aware of the different file movement configurations. Maintenance tasks are also discussed, such as: monitoring a running system, load balancing, high-availability set-up, and database tuning for performance.

Content

- SimManager Product Overview and Architecture
- Pre-Installation Planning
- SimManager Installation and Out Of the Box portal deployment
- Testing the Installation
- SimManager Directory Structure
- SimManager Studio Overview
- Database and Vault Administration
- Backup and Recovery
- Authentication
- Queuing Systems
- File Transfer Configuration
- High-availability, Load Balancing, and DB Connection Pooling
- System monitoring and tuning

Prerequisites SMM101-IntroductiontoSimManager

SMM102 - SimManager Basic Configuration

Duration 2 Days

Price $950

View Schedule

Simufact

SMF101 - Introduction to Simufact Forming

Objective

SMX101 is the introductory course for new SimXpert users. Students will master the basic skills required to use SimXpert in typical MCAE applications. They will be introduced to the Structures, Motion, Thermal, and Crash Workspaces. A brief introduction to building and running templates is also included. SMX101 emphasizes practical skills development through comprehensive, hands-on laboratory sessions. Students will learn to build analysis models using SimXpert, define material properties, create boundary conditions, apply loads, and submit their job for analysis and post-process results.

Content

- Take a model through the complete design and analysis cycle
- Learn graphics manipulation, imaging, and viewing techniques
- Import CAD geometry and manipulate it in SimXpert
- Practice using the different meshing techniques including morphing
- Learn how to verify and check the quality of a finite element model
- Study various ways to create Connections
- Nonlinear analysis of a bolt model
- Integration of Motion and Structures Workspaces
- Overview of Crash and Thermal Workspaces
- Macro creation
- Running and Building Templates

Prerequisites A basic knowledge of statics and strength of materials is recommended.

Duration 5 Days

Price $2,375

View Schedule

SMF102 - Simulation of Mechanical Joining Processes using Simufact Forming

Objective

SimXpert and MSC Nastran can be efficiently and effectively used to model laminate composites. The user can specify the material property, orientation, and thickness for each lamina in a composite layup. The program then calculates the properties of the equivalent plate. This automatic representation of laminated composites is available in most MSC Nastran solution sequences—including stresses, strains, and failure indices. In this class, SimXpert is used for pre- and post-processing.

MSC Nastran also supports 1D, 2D and 3D composite. In addition, MSC Nastran supports advanced composite failure methods such as Progressive Failure Analysis (PFA), Virtual Crack Closure Technique (VCCT), and Cohesive Zone Modeling (CZM).

Content

- Introduction to Composites in SimXpert and MSC Nastran
- Overview of Classical Lamination Theory
- Composite Beam
- Solid Composite, Solid Shell
- Composite Post-processing and Failure Theories
- Advanced Failure Theory and Prediction
- Interlaminar Shear
- Nonlinear Composite Analysis
- Buckling Analysis
- Practical Usage Guidelines

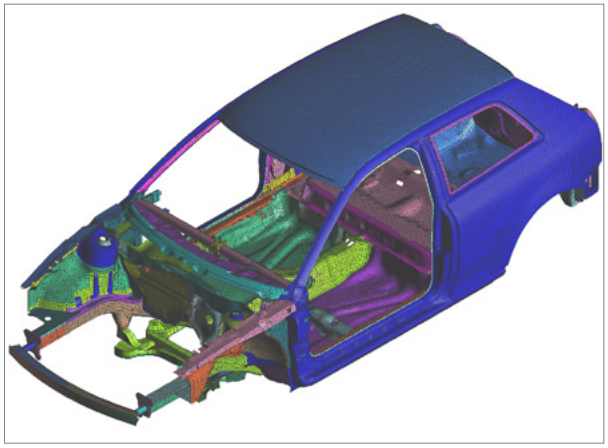
Prerequisites SMX120 (Linear Statics, Normal Modes, and Buckling Analysis using SimXpert), NAS101A (Linear Static and Normal Modes Analysis using MSC Nastran), NAS120 (Linear Static Analysis using MSC Nastran and Patran), or equivalent experience.

Duration 3 Days

Price $1,425

View Schedule

SMF201 - Introduction to Simufact Welding



Objective

SMX120 is the introductory course for the SimXpert Structures Workspace. Students will master the basic skills required to use SimXpert in typical Structural applications. SMX120 emphasizes practical skills development through comprehensive, hands-on laboratory sessions. Students will learn to build analysis models using SimXpert, define material properties, create boundary conditions, apply loads, and submit their job for analysis and post-process results.

Content

- Take a model through the complete design and analysis cycle
- Learn to import and edit Nastran input files
- Examine uses of different element types
- Become familiar with the contents of the Nastran output files
- Learn techniques to assess mesh quality
- Overview material types, including composites
- Investigate contact

Prerequisites A basic knowledge of statics and strength of materials is recommended. Although not required, a working knowledge of SimXpert such as that provided in the SMX101 class is recommended.

Duration 5 Days

Price $2,375

View Schedule

SimXpert

SMX101 - Introduction to SimXpert FE Workspaces

Objective

SMX101 is the introductory course for new SimXpert users. Students will master the basic skills required to use SimXpert in typical MCAE applications. They will be introduced to the Structures, Motion, Thermal, and Crash Workspaces. A brief introduction to building and running templates is also included. SMX101 emphasizes practical skills development through comprehensive, hands-on laboratory sessions. Students will learn to build analysis models using SimXpert, define material properties, create boundary conditions, apply loads, and submit their job for analysis and post-process results.

Content

- Take a model through the complete design and analysis cycle
- Learn graphics manipulation, imaging, and viewing techniques
- Import CAD geometry and manipulate it in SimXpert
- Practice using the different meshing techniques including morphing
- Learn how to verify and check the quality of a finite element model
- Study various ways to create Connections
- Nonlinear analysis of a bolt model
- Integration of Motion and Structures Workspaces
- Overview of Crash and Thermal Workspaces
- Macro creation
- Running and Building Templates

Prerequisites A basic knowledge of statics and strength of materials is recommended.

Duration 5 Days

Price $2,375

View Schedule

SMX113 - Composite Material Analysis using SimXpert

Objective

SimXpert and MSC Nastran can be efficiently and effectively used to model laminate composites. The user can specify the material property, orientation, and thickness for each lamina in a composite layup. The program then calculates the properties of the equivalent plate. This automatic representation of laminated composites is available in most MSC Nastran solution sequences—including stresses, strains, and failure indices. In this class, SimXpert is used for pre- and post-processing.

MSC Nastran also supports 1D, 2D and 3D composite. In addition, MSC Nastran supports advanced composite failure methods such as Progressive Failure Analysis (PFA), Virtual Crack Closure Technique (VCCT), and Cohesive Zone Modeling (CZM).

Content

- Introduction to Composites in SimXpert and MSC Nastran
- Overview of Classical Lamination Theory
- Composite Beam
- Solid Composite, Solid Shell
- Composite Post-processing and Failure Theories
- Advanced Failure Theory and Prediction
- Interlaminar Shear
- Nonlinear Composite Analysis
- Buckling Analysis
- Practical Usage Guidelines

Prerequisites SMX120 (Linear Statics, Normal Modes, and Buckling Analysis using SimXpert), NAS101A (Linear Static and Normal Modes Analysis using MSC Nastran), NAS120 (Linear Static Analysis using MSC Nastran and Patran), or equivalent experience.

Duration 3 Days

Price $1,425

View Schedule

SMX120 - Linear Statics, Normal Modes, and Buckling Analysis using SimXpert

Objective

SMX120 is the introductory course for the SimXpert Structures Workspace. Students will master the basic skills required to use SimXpert in typical Structural applications. SMX120 emphasizes practical skills development through comprehensive, hands-on laboratory sessions. Students will learn to build analysis models using SimXpert, define material properties, create boundary conditions, apply loads, and submit their job for analysis and post-process results.

Content

- Take a model through the complete design and analysis cycle
- Learn to import and edit Nastran input files
- Examine uses of different element types
- Become familiar with the contents of the Nastran output files
- Learn techniques to assess mesh quality
- Overview material types, including composites
- Investigate contact

Prerequisites A basic knowledge of statics and strength of materials is recommended. Although not required, a working knowledge of SimXpert such as that provided in the SMX101 class is recommended.

Duration 5 Days

Price $2,375

View Schedule

SMX121 - Complete Multibody Dynamics Analysis with SimXpert Motion

Objective

SMX121 is an introductory course in motion simulation using the Motion Workspace in SimXpert. Students will prepare motion models utilizing CAD geometry in SimXpert. Students will learn to create connections between parts, model contact and friction, model forces, and apply specific motion to parts in order to create motion models. Students will also learn to review motion results from these models using animations and charts.

Finally, the course demonstrates the modeling of flexible bodies in a motion simulation through interaction with the SimXpert Structures Workspace.

Content

- Import of CAD models into SimXpert
- Parts in SimXpert Motion Workspace
- Result Animations
- Charting Result Data
- Connections between motion parts
- Contact
- Forces, including spring dampers
- Applied Motion
- Flexible Body Simulation with Structures Workspace

Prerequisites No prior familiarity with SimXpert is required.

Duration 3 Days

Price $1,425

View Schedule

SMX122 - Dynamic Analysis using SimXpert

Objective

SMX122 is an introductory course in vibration simulation using the Structures Workspace in SimXpert. Students will prepare models utilizing imported geometry and geometry created in SimXpert. Students will learn to mesh and manually create elements, to create properties, time and frequency dependent loading, boundary conditions, and simulation conditions. Students will also learn to review results from these models using plots and charts.

Content

- Review systems with different Degrees of Freedom and their approach for solving them
- Understand the theory of frequency modes with respect to Degrees of Freedom and how to extract eigenvalues
- Investigate the effects of mesh size on frequency results
- Discuss different types of masses, how they differ, and when to use them
- Discover Reduction methods available with MSC Nastran
- Discuss Rigid Bodies, Response types, and Damping theory
- Introduce Modal and Direct Transient Response Analyses
- Introduce Modal and Direct Frequency Response Analyses
- Discuss the benefits of the Residual Vector Method
- Review Enforced Motion and the effect in Transient Analysis
- Investigate the theories and methods of Complex Modal Analyses
- Use the large mass method with transient response analysis
- Use the Effective Mass command to observe the mass participation

Prerequisites None

Duration 4 Days

Price $1,900

View Schedule

SMX124 - Thermal Analysis (SOL400) using SimXpert

Objective

SMX124 is an introductory course in heat transfer simulation using the Structural Workspace in SimXpert. Students will prepare thermal models utilizing imported MSC Nastran finite element models, geometry created in SimXpert, and hex mesh created by sweeping quad elements.

Students will learn how to create a temperature constraint, thermal heat flux LBC, internal heat generation LBC, free convection LBC, radiation to space LBC, radiation enclosure, variable direction solar LBC, and transient thermal result. Students will also learn to review thermal results for these thermal models using State Plot and Chart.

Content

- Overview of thermal analysis in SimXpert
- Topics on conduction
  - Theory, briefly
  - Creation of 1D, 2D, and 3D elements
  - Material for conduction
  - Temperature constraint
  - Initialization temperature
  - Loading: heat flux, and volumetric heat
- Analysis set-up for steady state analysis, including nonlinear parameters
- Display of steady state results
- Topics on convection
  - Theory, briefly
  - Free convection
- Topics on transient analysis
  - Transient temperature constraint
  - Transient loading
  - Transient convection
  - Initialization temperature
  - Transient analysis set-up, including nonlinear parameters
  - Display of transient results
- Topics on radiation
  - Theory, briefly
  - Radiation to space
  - Radiation enclosures
  - Vector flux from a distant source
- Thermal analysis theory
  - In more depth
  - Nonlinear parameters
  - Recommendations

- Import of MSC Nastran models into SimXpert
- Import of CAD models into SimXpert
- Parts in SimXpert

Prerequisites None

Duration 3 Days

Price $1,425

View Schedule

SMX126 - Explicit Nonlinear Analysis using SimXpert

Objective

SMX126 is an introductory course in dynamics simulation using the Explicit Workspace in SimXpert. Students will prepare short duration structural dynamics models utilizing the imported MSC Nastran finite element models.

Students will learn how to create and/or modify material and part properties, loads, and boundary and initial conditions for dynamic simulation models. They will also learn how to set up the jobs for running the models with the Explicit solver, and review the results for these models using State Plot and Chart.

Content

- Overview of Explicit Dynamic Analysis in SimXpert
- Overview of SimXpert
- Theory, briefly
- Import of MSC Nastran models into SimXpert
- Parts in SimXpert Explicit workspace
- Material Models for Explicit Dynamic Analysis
- Elements in Explicit
- Loads and Boundary Conditions
- Contact Bodies and Contact Properties
- Initial Conditions
- Selection of Time Steps
- Explicit Analysis set-up, including nonlinear parameters
- Display of Explicit Analysis Results
- Hands on Workshops

Prerequisites No prior familiarity with SimXpert is required

Duration 3 Days

Price $1,425

View Schedule

SMX131 - SimXpert Templates

Objective

SMX131 is a course that focuses on the SimXpert Template Builder workspace. Students will gain the skills to create simulation templates that automate complex and/or repetitive analysis procedures. In the SimXpert Template Builder learners will build templates that utilize several different workspaces and test them in the Template Execution environment. Students will also be able to create actions, the basic building blocks of templates, and use the various connection, branching, and looping tools to add complexity to a template. Publishing templates to the enterprise via the SimManager portal is also explored.

Content

- Explore the Template Builder workspace
- Run a simple Structures template in Template Execution window
- Build a simple template using built-in actions and connection tool
- Record a macro (template) and then edit it in Template Builder
- Build a template containing choice and loop tools
- Create a new embedded script action (using RADE language)
- Create a new core script action (using Python language)
- Execute some additional templates and explore various levels of automation
- Learn how publishing and retrieving templates via the SimManager portal allows sharing across the enterprise

Prerequisites SMX101 (Introduction to SimXpert FE Workspaces) is recommended before taking SMX131 (SimXpert Templates).

Duration 3 Days

Price $1,425

View Schedule

SMX132- SimXpert Scripting

Objective

In this course students will learn to write scripts in the Python scripting language. They will create their own custom Template Builder actions. They will learn to use classes and functions in the SimXpert Application Programming Interface (API) to access, modify, and create objects in the SimXpert database. After completing this course students will understand the SimXpert data model and the major classes in the API and be able to navigate the API documentation to find the appropriate classes and functions for their particular purpose. They will know how to modify the SimXpert menus and tool ribbon to display custom actions and scripts. They will create and use Expressions to populate input fields by invoking an external action.

Content

- Overview of Python scripting language
- Review / overview of the Template Builder workspace
- Using Script Editor and Debugger tool
- Programming approaches
- Using the SimXpert Application Programming Interface (API) to obtain data from a model and to create and modify entities
  - Graphics objects
  - Finite element modeling
  - Materials and Properties API
  - Loads and Boundary Conditions API
  - Geometry
  - Analysis Setup and Submittal
  - Results post-processing
  - Parts, Groups, and Sets
- Different ways to invoke Actions in SimXpert
- Creating and using Expressions
- Creating and using Custom Attributes

Prerequisites SMX131 (SimXpert Templates) is recommended before taking SMX132. SMX132 (SimXpert Scripting) is a course for advanced users with a programming background. Knowledge of Python is helpful.

Duration 3 Days

Price $1,425

View Schedule

SMX140 - Dynamic System Modeling and Simulation using SimXpert

Objective

This course gives the engineer a basic understanding of the SimXpert Systems and Controls Workspace modeling, simulation, and analysis environment. Students learn to represent an existing dynamic system on screen using predefined modeling blocks and components, empirical table data, and user-defined components. Students learn to calculate stable operating points using the Steady-State Analysis, and are introduced to fixed- and variable-step integration used during simulation. The class also covers linear analyses, including, in part, transfer function calculation, frequency response generation, and root locus analysis.

Content

- Model Building
- Linear Analysis
- Analysis Tool and Methodology
- Simulation and Integration
- Modeling Discontinuities and Switch States
- Data Tables and the Matrix Editor
- Operating Point and Steady State Analysis
- Working with User Code Components
- Discrete Components and States
- Writing Code in Systems and Controls Workspace
- Matrix Algebra Tool

Prerequisites An engineering background, knowledge of FORTRAN, and an understanding of differential equations.

Duration 2 Days

Price $950

View Schedule

SMX400 - Nonlinear Structural Analysis using SimXpert

Objective

In this class, you will be exposed to various aspects of implicit nonlinear analysis using SimXpert. This includes large deformation, advanced nonlinear material, contact, analysis chaining, heat transfer, and nonlinear transient dynamics.

Content

- Nonlinear versus Linear Analysis
- Iterative Solution Methods
- Advanced Nonlinear Materials
- Nonlinear Elements
- Contact Analysis
- Heat Transfer
- Nonlinear Transient Dynamics

Prerequisites SMX120 (Linear Statics, Normal Modes, and Buckling Analysis using SimXpert) or equivalent.

Duration 3 Days

Price $1,425

View Schedule

SMX702 - Fundamentals of Multibody Dynamics Analysis with SimXpert

Objective

This course is intended for users who need a basic understanding of the SimXpert Motion workspace. Presented in the class are all of the basics of building models in SimXpert Motion (PARTs, JOINTs, MOTIONs, forces, function expressions, simulation types), running simulations and simple plotting. Users who intend to do anything more than simple model creation are encouraged to take the 3-day SMX121 (Complete Multibody Dynamics Analysis with SimXpert Motion) class instead of this one.

Content

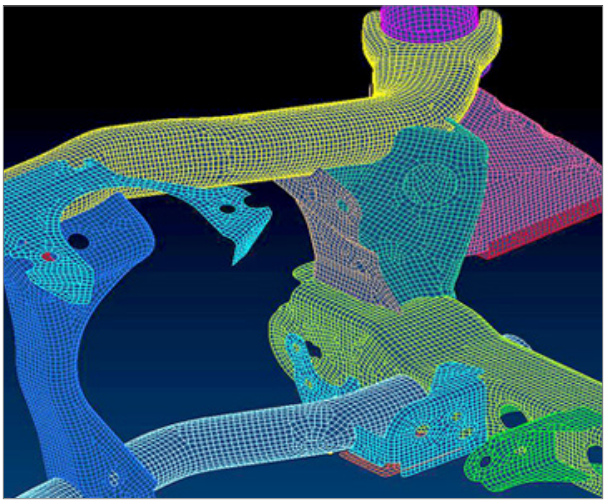
- Building models
  - Adams dataset structure (.adm)
    - Statements
    - Functions
  - Connecting parts with:
    - Ideal constraints (hinges, sliders, etc.)
    - Flexible connectors (springs, dampers, etc.)
    - Complex forces based on system states or test data
    - Contacts using IMPACT function
  - Actuating a system with:
    - Motions
    - Applied forces
    - Gravity
  - Measuring quantities of interest
    - Displacements
    - Velocities
    - Accelerations
    - Forces
- Simulating models
  - Adams command files (.acf)
  - Compare/contrast simulation types
    - Dynamic
    - Kinematic
    - Quasi-static
    - Static
- Reviewing results
  - Plotting/Animating(.msg)
  - Message Files (.msg)
  - Analysis Files (.gra, .res, .req, .out)

Prerequisites None

Duration 2 Days

Price $1,425

View Schedule



Sinda

SND501 - Network Thermal Analysis Using MSC Sinda

Objective

The intent of this course is to provide users with comprehensive exposure to the MSC Sinda thermal analysis program.

Content

- Day 1 & 2 - Thermal Modeling Using MSC Sinda: The intent of this workshop is to provide users with comprehensive exposure to the MSC Sinda thermal analysis program. A combination of lecture and hands-on instruction is used to teach how to create MSC Sinda thermal models involving real world thermal boundary conditions.
- Day 3 - Numerical Methods in MSC Sinda: The finite difference numerical methods used in the steady state and transient solutions of MSC Sinda will be discussed. Information on convergence, stability, control constants, and errors associated with the numerical methods will be presented.
- Day 4 - Advanced MSC Sinda: This day cover advanced MSC Sinda concepts. The flow of a steady state or transient model is discussed. The Fortran file produced by MSC Sinda is discussed so all key variables and arrays are described. Adding Fortran and using these variables and arrays are also addressed. The Pre-Preprocess (PPP mode) is explained and lab problems with global variables will be done.
- Day 5 – MSC Sinda for PATRAN and other Graphical Modelers: MSC Sinda Covers the basics of using our graphical pre-post processors such as MSC Sinda for PATRAN. Participants will learn how to create geometry in 3-D, add boundary conditions, and post process the results.

Prerequisites None

Duration 5 Days

Price $2,375

View Schedule

SND502 - Thermal Analysis Using Patran with MSC Sinda

Objective

This course will allow users to explore some of the more in-depth features of creating advanced MSC Sinda thermal models from a finite element model builder. Students will use Patran to build the lab problems.

Content

- The following are some topics that will be covered:
  - Using MSC Sinda skeleton files
  - Adding FORTRAN to graphically built models
  - Creating groups
  - Excel material files
  - Advanced mesh control
  - Modeling time and temperature dependent properties
  - Advanced geometry manipulation
  - Report generation - using the new Thermal Studio
  - Manipulating models and views
  - Importing and running large models
  - Adding contacts between surfaces
  - Importing, simplifying and using CAD geometry
  - Exploring materials and properties
  - Radiation super elements and primitives - for fast and accurate radiation modeling
  - Radiation enclosures
  - Special features and applications
  - View Scheduling to radiation codes such as THERMICA, Nevada, TRASYS, and TSS
  - Using thermal data for stress/distress analysis

Prerequisites None

Duration 3 Days

Price $1,425

View Schedule

SND503 - Spacecraft Thermal Analysis Using THERMICA V4

Objective

Thermica was designed by spacecraft thermal engineers and is one of the most comprehensive spacecraft thermal design tools available in the industry today. Whether you are currently using Thermica or other software for your spacecraft thermal design, you will gain information about these new tools that could increase your productivity. Thermica is a comprehensive spacecraft thermal design system that transparently incorporates the MSC Sinda thermal analyzer into a powerful state-of-the-art graphical user environment. Thermica version 4 is a major upgrade to the previous versions of Thermica that have been used for more than two decades on dozens of major space programs.

V4 has a completely new GUI that runs in Windows, Linux and other workstations as native code and graphics. This new user interface has a Windows XP look and feel and is easy and intuitive to learn. The new model builder has View Schedules to CAD and Nastran and a powerful new shape based meshing tool. Photo-like animations of spacecrafts traveling to and orbiting other texture mapped planets can be viewed interactively and recorded as movie files.

Results such as temperature, heat fluxes and view factors can be graphically visualized on these animations and also in interactive 2D plots. Powerful kinematic motion such as deployment of solar arrays and arbitrary moving surfaces can be modeled. Missions to the moon, Mars and other planets can be completely simulated using time-position-velocity data from external codes such as Satellite Tool Kit (STK).

Content

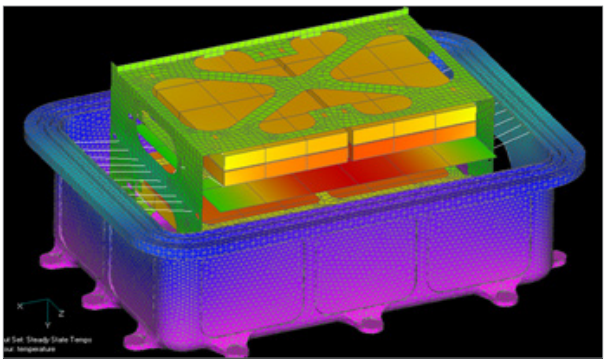
- Day 1 – Overall Process Flow of Thermica V4 and Model Building
  - Understanding the process flow of V4 from model building, meshing, orbit setup, kinematic motion, mission setup, Sinda model generation and basic post processing.
  - Model building process in V4.
  - Using CAD files for building models.
  - Using Nastran files for model building and exporting Thermica models to Nastran.
- Day 2 – Orbit Setup, Kinematic motion and Mission Setup
  - Orbit definition including using external data from STK to define these.
  - Kinematic motions with moving bodies and arbitrary motions such as solar panels deploying.
  - Mission setup where a model, mesh, orbit and motion are put together. Real time animations and mission viewing options will also be explored.
- Day 3 – Sinda model creation, post processing and advanced features
  - Computing radiation exchange factors and orbital heating.
  - Generation of a Sinda thermal model.
  - Accuracy checks on ray tracing runs.
  - Post processing of temperatures, view factors and orbital heating using 3D animations of the spacecraft and planets.
  - Recording animations as standard or High Definition (HD) movies.
  - 2D interactive plotting of temperatures, view factors and orbital heating

Prerequisites None

Duration 3 Days

Price $1,425

View Schedule



Contacts

Offices

Corporate

MSC Software Corporation
4675 MacArthur Court
Newport Beach, California 92660
Telephone +1 (800) 732 7211
msc.training@mscsoftware.com

Europe, Middle East, Africa

MSC Software GmbH
Am Moosfeld 13
81829 Munich, Germany
Telephone +49 (89) 21093224 Ext. 4950
Please dial extension only after voice request!
info.europe@mscsoftware.com

Asia-Pacific

MSC Software (S) Pte. Ltd.
The Concourse
300 Beach Road, Unit #30-06
Singapore 199555
Telephone +65 (6272) 0082
apac.contact@mscsoftware.com

Latin America

MSC Software Brasil Ltda.
Rua Augusta, 1598 - Cj. 61
01304-001, Sao Paulo - SP, BRASIL
Main Phone: +55 (11) 3146-0277
Fax Phone: +55 (11) 3146-0270
marketing.mscbrasil@mscsoftware.com

Japan

MSC Software Japan LTD.
Shinjuku First West 8F
23-7 Nishi Shinjuku
1-Chome, Shinjuku-Ku
Tokyo, Japan 160-0023
Telephone +81 (3) 6911 1200
mscj.market@mscsoftware.com

India

MSC Software Corporation India Pvt. Ltd.
5AC, 961, 1st Block
HRBR
Bangalore, Karnataka, India 560 043
Telephone +91 (80) 4091 7449
Fax +91 (80) 4091 7446
contact.india@mscsoftware.com

Please visit www.mscsoftware.com/locations for a map of all MSC locations.

TC*2017OCT*BR

MSC Software