HyperWorks®

A Platform for Innovation™

Altair® HyperWorks®
Training Brochure

HyperWorks is a division of Altair

HyperWorks

Our goal at Altair Engineering is to offer a training curriculum that improves the productivity of our customers. The objective is to deliver an excellent source of information that is both easy to use and applicable to real life problems.

Classes are offered at our regional offices as well as at some of our partner universities. The training manuals provided to each student contain chapter notes and hands-on exercises. Models for each exercise are available at the end of each class.

The course overviews in this brochure provide detailed descriptions of the courses offered by Altair. The courses range from an Introduction to Finite Element Analysis to advanced topics using the HyperWorks suite of tools.

We are committed to providing training material that is best-in-class and look forward to the opportunity to pass this knowledge to our customers.

HyperWorks®

CONTENTS

PRE AND POST-PROCESSING

SOLVER AND OPTIMIZATION

MULTIBODY DYNAMICS

SCRIPTING

METAL FORMING

HyperMesh Introduction

Pre-processing for Finite Element Analysis

This is an introductory course for using HyperMesh to create and set up finite element models for analysis. A combination of lectures and exercises will familiarize students with the HyperMesh environment, process, and suite of tools needed to start using HyperMesh. Lessons are taught with a process driven approach that challenge the student to learn, use and explore the software in real world situations. This class is usually combined with the HyperView Introduction class as a 3 day training package.

Class Format

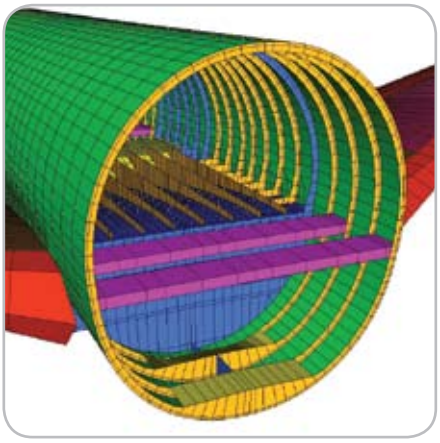
- 2 ½ days
- Instructor led
- Presentation and hands-on exercises
- One student to a computer

Prerequisites

A basic knowledge of finite element analysis (recommended)

Topics

- Basic interaction with HyperMesh
  - User interface
  - Opening / saving files
  - Working with panels
  - Model organization
  - Display control
- Preparing geometry for meshing
  - Repairing surface topology
  - Midsurfaces
  - Defeaturing models
- Shell meshing
  - Refining surface topology
  - Automeshing – meshing on surface geometry
  - Checking and editing mesh
  - Batch meshing
- Creating hexa and penta mesh
  - Creating & editing solid geometry
  - Creating hex mesh with the solid map panel
- Tetra meshing
  - Volume TetraMeshing
  - Standard TetraMeshing
  - TetraMesh Process Manager
- 1D meshing
  - Rigid Element Creation
  - Bar and Beam Elements



HyperView Introduction

Results Visualization and Data Analysis for Finite Element Analysis

This is an introductory course for using HyperView to post-process finite element solver results and to analyze test lab engineering data. This course is normally taught in conjunction with HyperMesh Introduction.

Class Format

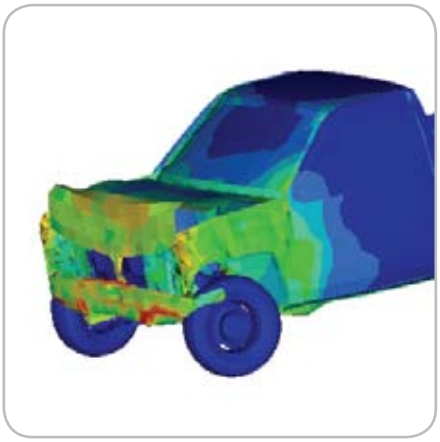
- 1 or 1/2 day (Usually taught in conjunction with HyperMesh Introduction)
- Instructor led
- Presentation and hands-on exercises
- One student to a computer

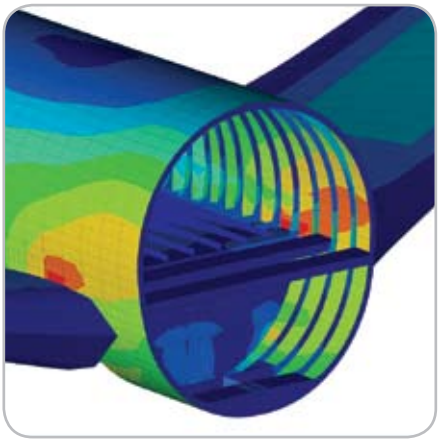
Prerequisites

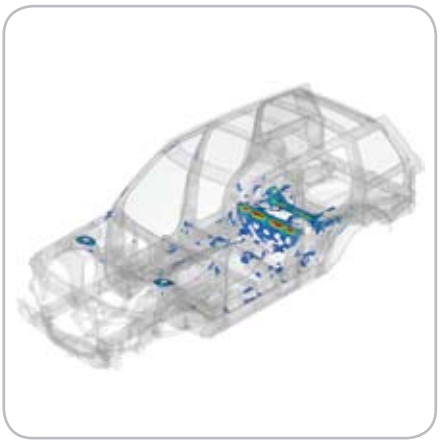
A basic knowledge of finite element analysis (recommended)

Topics

- Basic interaction with HyperView
- Animating results in the animation window
- Results visualization: contour, deformed, iso surface, tensor and vector plots
- Results manipulation: tracing, tracking, querying, measuring data, and creating section cuts, notes, and derived load cases
- Creating XY curves using data files and math functions
- Modifying XY plots and curve display attributes
- Interrogating XY curves
- Curve math
- Translating results
- Presenting Results
  - Creating BMP, JPEG, TIF, AVI files and HTML files
  - Visualizing and sharing CAE results over the Internet using the freely available HyperView Player
  - Publishing post-processing sessions in HTML and Microsoft PowerPoint XML formats.



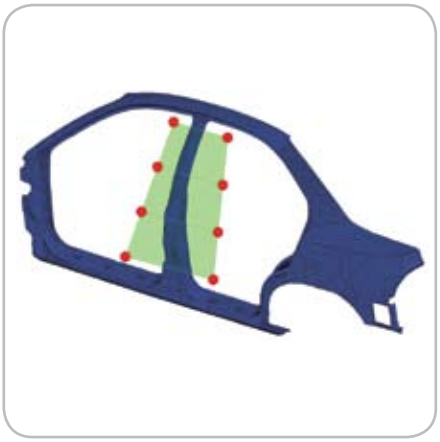




HyperMorph Introduction

Methods for Morphing Finite Element Models

HyperMorph is a mesh morphing tool that allows you to alter finite element models while keeping mesh distortions to a minimum. After morphing has been performed, you can visualize the quality of the mesh, and can automatically smooth it if need be. A re-mesh can also be performed, keeping the morphing entities like handles, domains and shapes intact.

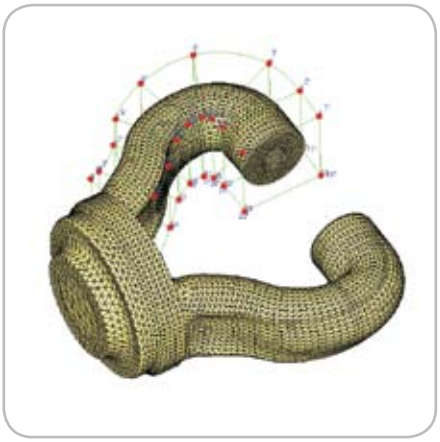


Class Format

- 1 day
- Instructor led
- Presentation and hands-on exercises
- One student to a computer

Prerequisites

A basic knowledge of HyperMesh. It is recommended students complete the HyperMesh and HyperView Introduction courses.



Topics

- Change the profile and the dimensions of your mesh
- Map an existing mesh onto a new geometry, and
- Create shape variables that can be used for optimization
- Freehand Morphing
- Map to Geometry
- Morph Volumes, Domains, and Handles
- Morphing constraints, Symmetries, and Biasing factors
- Morphs can be saved as Shapes
- Animated to review the morphing
- Transfer loads from one model to another

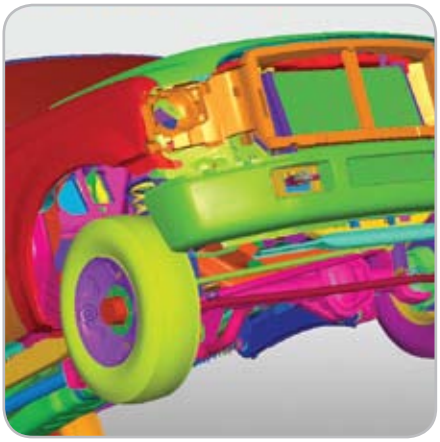
HyperCrash Introduction

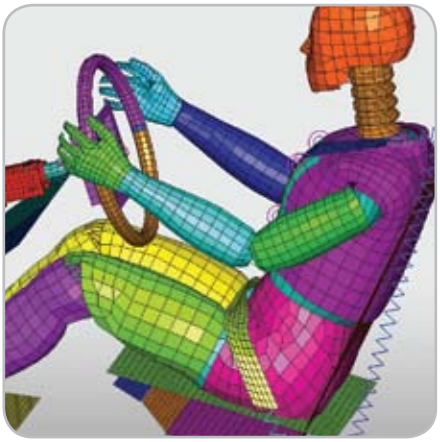
Preprocessing for RADIOSS Crash Analysis

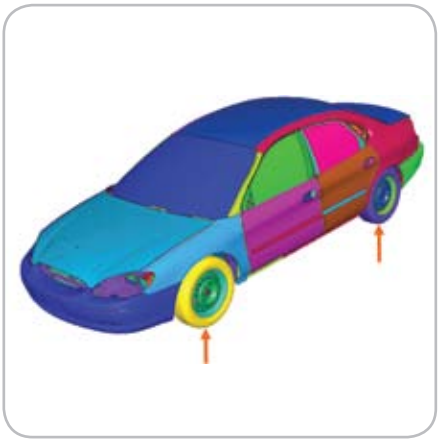
HyperCrash is a pre-process for RADIOSS which is tailored to meet the needs of automotive crash users. HyperCrash enables the users to build the highest quality model, with significant decrease in modeling time, and with highest level of homogeneity.

Also, HyperCrash has an Automotive Safety module which consists of the following tools: Dummy Positioner, SeatBelt Generator, Airbag Folder and Seat Deformer.

HyperCrash has various quality checks, including it most powerful intersection/penetration checking and fixing routine, which enables the users to setup a model that is perfectly consistent with the RADIOSS solver. HyperCrash has built-in automated routines that allow the users to significantly reduce the modeling time. Also, it allows the users in a specific group to build a homogeneous model by allowing these users to access the same databases like materials, properties, spotweld connections, dummies, barriers, same checks and etc.







Class Format

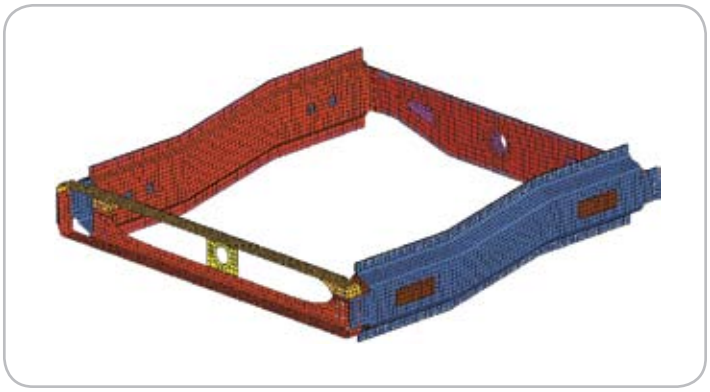
- 1 day
- Instructor led
- Presentation and hands-on exercises
- One student to a computer

Prerequisites

None

Topics

- Managing Tree Hierarchy
- Materials & Properties Creation and Allocation
- Element Modifications
- Connections
- Merging & Connecting Sub-Systems
- Loadcase & Interfaces Definition
- Quality Checks
- Usage of Replace Feature
- Time History Selection
- Cleaning & Exporting the Model
- Safety: Dummy Positioner, Seat Belt Generator
- Also, Airbag Folder, Seat Deformer, and Seatbelt Generator could be demonstrated



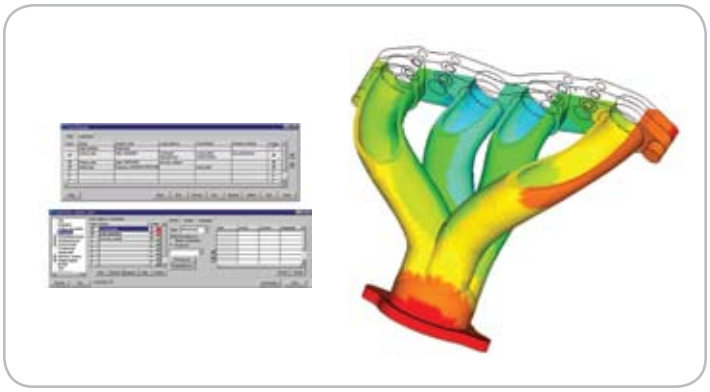
HyperWorks for LS-DYNA

This course helps students learn how to use HyperMesh to create and interface with LS-DYNA formatted models and to use HyperView to post-process LS-DYNA results.

Class Format
- 1 day
- Instructor led
- Presentation and hands-on exercises
- One student to a computer

Prerequisites
None

Topics
- Basic interfacing with HyperMesh
- Defining model data
- Defining boundary conditions and loads
- Defining control cards and specifying output
- Use of beams, rigid bodies, joints, and loads
- Model importing
- Rigid wall, model data, constraints, and outputs
- Assemblies
- Interfacing for Arbitrary-Langrangian-Eulerian capability



HyperWorks for ABAQUS

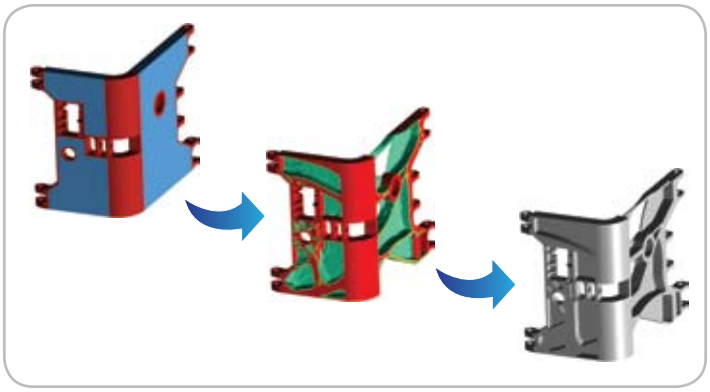
Interfacing with ABAQUS using HyperMesh and HyperView

This course helps students learn how to use HyperMesh to create and interface with ABAQUS formatted models and to use HyperView to post-process ABAQUS results.

Class Format
- 1 day
- Instructor led
- Presentation and hands-on exercises
- One student to a computer

Prerequisites
- *A basic knowledge of ABAQUS*
- *A basic knowledge of HyperMesh and HyperView. It is recommended students complete HyperMesh Introduction*

Topics
- Interfacing with HyperMesh
- Organization of ABAQUS input files
- Review and modify ABAQUS entities
- Define section properties and materials
- Relate ABAQUS entities to HyperMesh element and load configurations
- Define kinematic constraints
- Define one node elements
- Define loads on geometry and map them to elements
- Define systems
- Define contacts using the Contact Manager
- Define loads for model data using the Step Manager
- Define history data, including *STEPs, loads, output requests, contacts (for ABAQUS\Explicit) using the Step Manager
- Create and import curves to define loading
- ABAQUS INP and ODB files in the animation window
- Creating contour plots for ABAQUS results
- Plotting history data from ABAQUS ODB files



OptiStruct Optimization

Analysis, Concept and Optimization

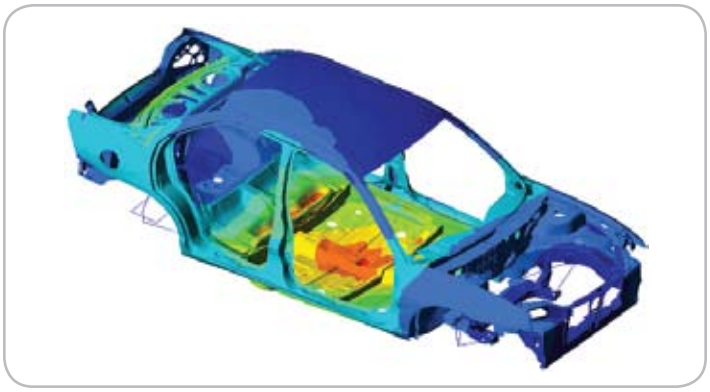
This is an introductory course teaching the use of OptiStruct for optimization in the product design process, this is a two day training with focus on understanding how optimization can help on improve the product structural performance. Students will learn optimization concepts and do hands-on exercises for topology, topography, size, and shape optimizations.

Class Format
- 2 days
- Instructor led
- Presentation and hands-on exercises
- One student to a computer

Prerequisites
- *HyperMesh Introduction*
- *Radioss Linear Introduction*

Topics
- HyperWorks Overview
- Optimization Theoretical Background
- HyperMesh Optimization Interface
- Concept Design: Topology, topography, Free-size optimization.
- Topology optimization with manufacturing constraints
- Geometric data generation of new design concept using OSSmooth which is part of OptiStruct
- Fine-Tuning: Shape, size and free-shape optimization

Suggested Follow-up Courses
- *HyperMorph Introduction*
- *HyperStudy Introduction*



RADIOSS for Linear Analysis

Static, Modal, Linear Buckling, and Inertia Relief

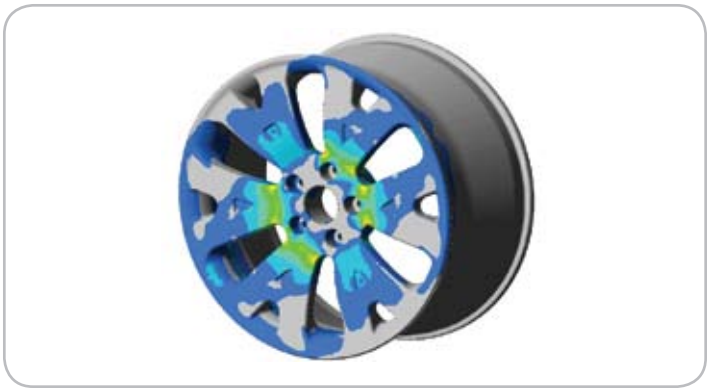
This is an introductory course for using RADIOSS to solve static linear problems. This is a one day training with a focus on understanding how to solve structural problems using a high level finite element solver. The theoretical and practical aspects of these problems are presented toprovide the attendees the knowledge to use RADIOSS on their own models.

Class Format
- 1 day
- Instructor-led
- Presentation and hands-on exercises
- One student to a computer

Prerequisites
- *A basic knowledge of finite element analysis*
- *A basic knowledge about Pre and Post processing finite element models.*

Topics
- HyperWorks Overview
- RADIOSS Integration with HyperWorks
- Linear Static Analysis
- Model Definition Structure
- Modal Analysis
- Natural frequency and Mode shape background
- Linear Buckling Analysis
- Inertia Relief Analysis

Suggested Follow-up Courses
- *HyperStudy Introduction*
- *OptiStruct Optimization*
- *RADIOSS Linear Dynamics*



RADIOSS for Linear Dynamics

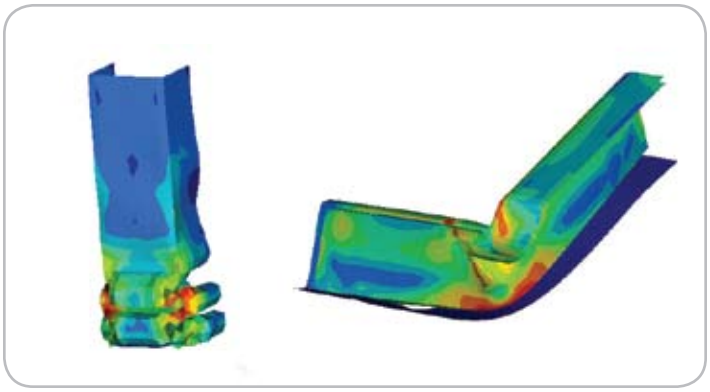
This is an advanced course in RADIOSS to solve problems where the dynamic forces are important. This is a two day training with focus on linear problems that are solved on the frequency domain using Forced Frequency Response (FRF) or on time domain using Transient Analysis. Students will learn the theoretical and practical concepts necessary to solve a structural problem with dynamic excitations.

Class Format
- 2 days
- Instructor-led
- Presentation and hands-on exercises
- One student to a computer

Prerequisites
RADIOSS Linear

Topics
- Modal Analysis
  - Natural Frequency
  - Mode Shape
  - Eigenvalue Solution
- Forced Frequency Response
  - Direct Frequency Response
  - Modal Frequency Response
- Transient
  - Transient Response Analysis
  - Direct Transient Response
  - Modal Transient Response

Suggested Follow-up Courses
- *HyperStudy*
- *OptiStruct Optimization*
- *RADIOSS Fatigue Analysis*



RADIOSS for Impact Introduction

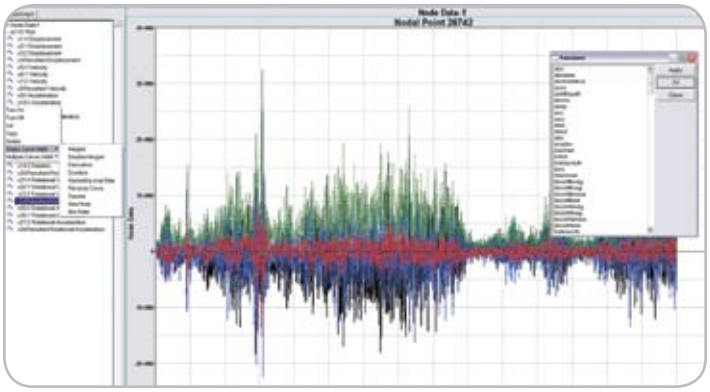
This is an introductory course for using RADIOSS to solve a large number of high non-linear dynamics problems, with large displacements, large strains, contact and material non-linearity. This is a two day training with a focus on understanding how to solve real problems using a high level finite element solver. The theoretical and practical aspects of these problems are presented to provide the attendees the ability to use RADIOSS on their own models.

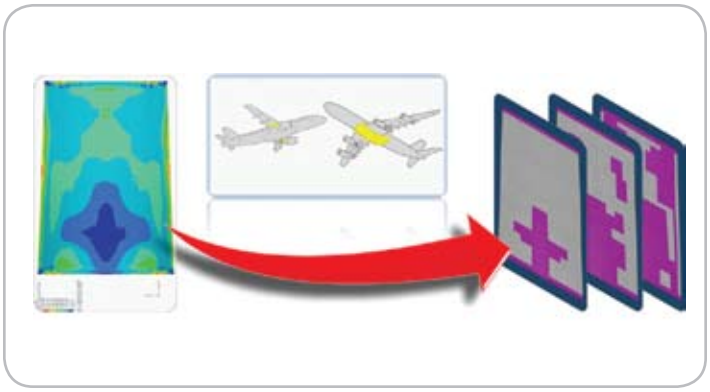
Class Format
- 2 days
- Instructor-led
- Presentation and hands-on exercises
- One student to a computer

Prerequisites
- *A basic knowledge of finite element analysis*
- *A basic knowledge of HyperMesh or HyperCrash*
- *A basic knowledge about Pre and Post processing finite element models*

Topics
- Background about finite element formulations and time integration schemes
- Pre and Post-processing structural dynamic simulations
- Time-step stability and control
- Contact interfaces
- Material Laws
- Debugging models and understanding error messages
- Best Practices on solving high non-linear problems

Suggested Follow-up Courses
- *HyperStudy*
- *OptiStruct Optimization*





RADIOSS for Fatigue Analysis

Stress and Strain Life Approach for High and Low Cycle Durability Designs

This is an introductory course using RADIOSS to solve problems where fatigue failure is possible. This is a one day training with focus on problems where the failure is well represented using SN or EN approach. The theoretical and practical concepts necessary to solve this problem are discussed and applied on practical exercises

Class Format
- 1 day
- Instructor-led
- Presentation and hands-on exercises
- One student to a computer

Prerequisites
Radioss Linear

Topics
- Fatigue Introduction
- Mean Stress Effect
- Fatigue Coefficients
- Load History
- Cycle Counting
- Linear Superposition of Multiple FEA/Load Time History
- Fatigue Results
- Damage Model
- Hysteresis Loop Shape
- Neuber Stress correction

Suggested Follow-up Courses
- *HyperStudy*
- *OptiStruct Optimization*
- *RADIOSS Linear Dynamics*

Composite Analysis

Analysis, Concept, and Optimization of Composite Structures

This course is intended to provide an overview of performing Composite Analysis using HyperMesh, OptiStruct and RADIOSS. Students will learn the basic concepts about composite and how to perform analysis and optimization on models with these materials. This is a theoretical and practical course that covers all design steps for systems manufactured with composite materials using an automated optimization technique developed inside HyperWorks.

Class Format
- 2 days
- Instructor-led
- Presentation and hands-on exercises
- One student to a computer

Prerequisites
Radioss Linear

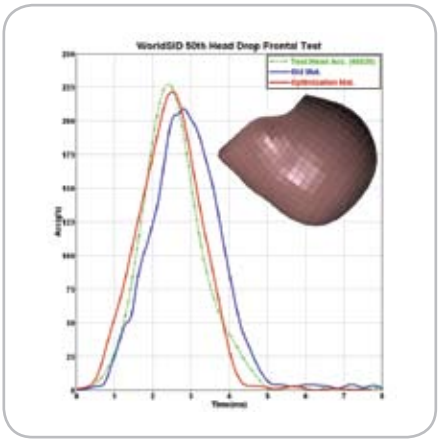
Topics
- Composite Material Definitions
- Laminated Plate Theory
- Predesign
- Design Analysis
- Composite setup using HyperMesh
- Optimization Techniques
- Topology Optimization
- Free-Size Optimization
- Concept Design Synthesis
- Size Optimization
- Size Optimization applied to Composites
- Ply Stacking Sequence Optimization

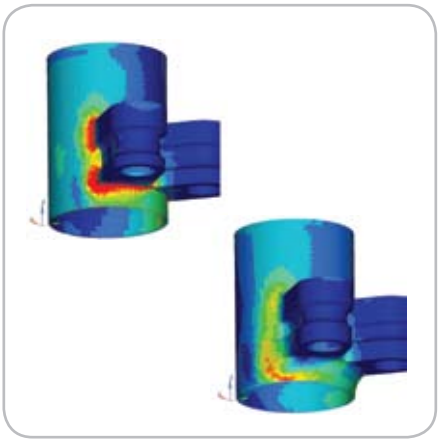
Suggested Follow-up Courses
- *HyperStudy*
- *OptiStruct Optimization*
- *RADIOSS Linear Dynamics*
- *RADIOSS Fatigue Analysis*

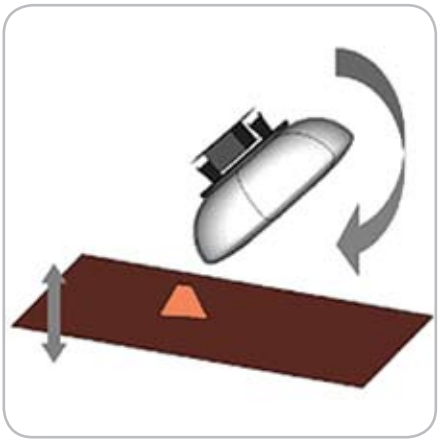
HyperStudy

DOE, Approximation, Optimization and Stochastic Studies

This is an introductory course teaching the use of HyperStudy to do design of experiments (DOE), optimization, and stochastic studies with various linear and non-linear solvers. Methods/techniques, applications, problem setup and post-processing are discussed for these study types.







Class Format

- 1 day
- Instructor led
- Hands-on exercises, one student to a computer

Prerequisites

- *A basic knowledge of finite element analysis*
- *Basic knowledge of HyperMesh*
- *HyperMesh Introduction (Recommended)*
- *A basic knowledge of HyperMorph which is a part of HyperMesh*
- *Morphing Introduction (Recommended)*

Topics

- Methods/Techniques and applications for DOE, approximations, optimization and stochastic studies
- Discussion of differences between OptiStruct, structural concept design and optimization and HyperStudy methods and applications
- Setup of optimization, stochastic and DOE models for various solvers such as Radioss, LS-DYNA and ABAQUS
- Generation of shape variables using HyperMesh and HyperMorph and their integration into HyperStudy
- Parameterization of solver input files using Altair Templex
- Addition of linear or non-linear solver executables or run scripts to HyperStudy
- Interpretation of results from optimization, stochasti and DOE studies

Suggested Follow-up Courses

- *OptiStruct Optimization*
- *RADIOSS Linear Dynamics*
- *RADIOSS Fatigue Analysis*

RADIOSS

For Defense Applications

The goal of this two day course is to an understanding of blast and explosion simulations using RADIOSS. The course will cover the theoretical background of blast and simulation topics as well as the practical use and setup using RADIOSS.

Class Format

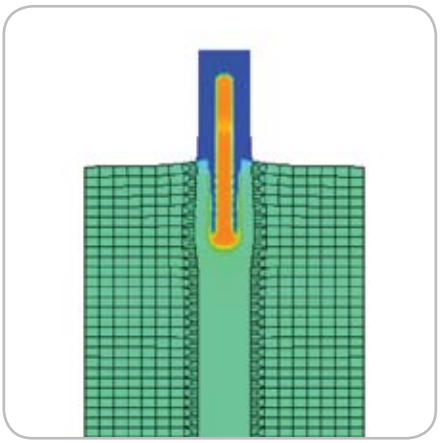
- 2 days
- Instructor led
- Presentation and hands-on exercises
- One student to a computer

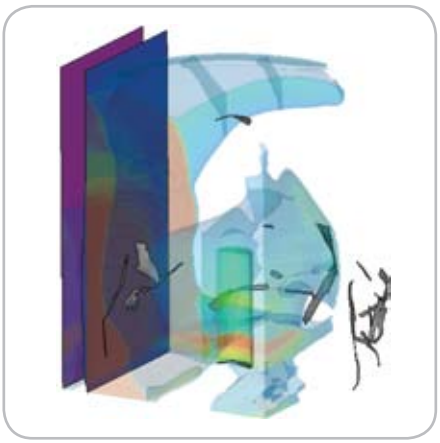
Prerequisites

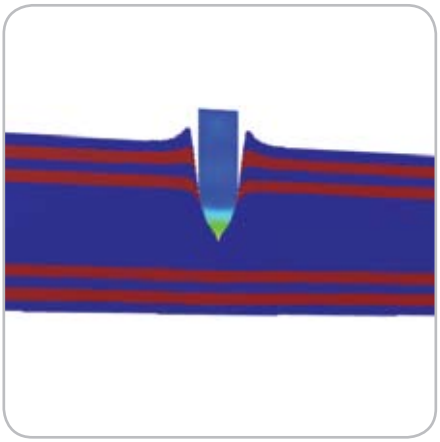
None

Topics

- Basic theoretical aspects
  - Algorithm
  - ALE formulations
- Solid elements in RADIOSS
- Recommandations for meshing
- Material laws
- Boundary conditions
- SPH
- SPH/ALE comparison
- Specific applications
  - Impact
    - Non Penetrating impact
    - Penetrating impact
  - Underwater explosion
    - Explosion modeling
    - Boundary conditions / surface modeling
    - Close field explosion
    - Far field explosion





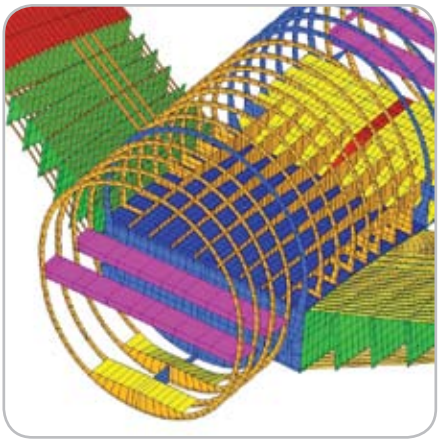


Basic FEA

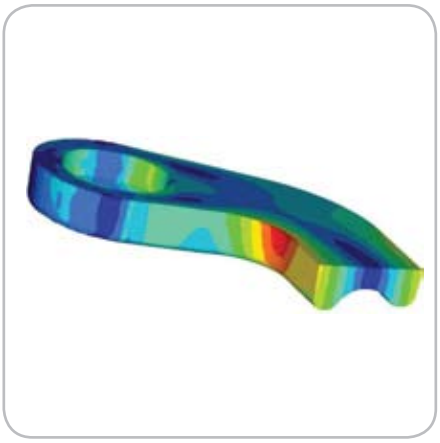
This course is intended to provide an introduction to the Finite Element Analysis (FEA) method for structural analysis. Students are expected to have a engineering background but they do not need to have experience with FEA. Students will learn the basic steps to set up an FEA job, the issues to consider in setting up a job, and how to interpret the results.

Class Format
- 3 days
- Instructor led
- Presentation and hands-on exercises, one student to a computer

Prerequisites
None



Topics
- Strategic Planning
  - Planning the Solution
  - Creating a Solution Checklist
  - Boundary Conditions and Load Cases
  - Linear Assumption
- The Art of Modeling
  - Consistent Units
  - Understanding Element Behavior
  - Element Selection
  - Mesh Density and Solution Convergence
  - Compatability and Mechanisms
  - Rigid Elements
  - Fasteners
- Analysis Types
  - Static Analysis
  - Stress Results
  - Dynamics
  - Buckling
  - Presentation of Results



Suggested Follow-up Courses
- *RADIOSS Linear Dynamics*
- *HyperStudy*
- *OptiStruct Optimization*

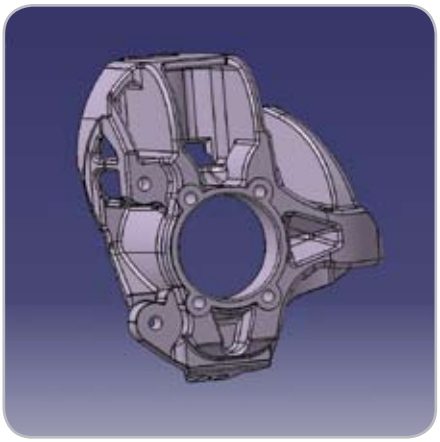
HyperShape/CATIA 3.1

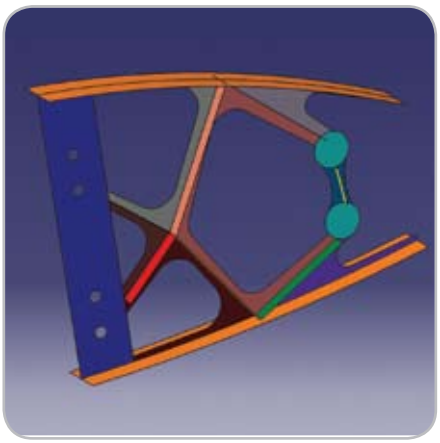
HyperShape/CATIA training class allows designers to learn how to get the most efficient design concepts without leaving the CATIA V5 environment. The designs that are produced are more robust and lighter.

Class Format
- 1 day
- Instructor Led
- Presntation and hands-on exercises
- One student to a computer

Prerequisites
Familiar with CATIA Analysis workbench

Topics
- Topology optimization
- Free-shape optimization
- Topography optimization
- Gauge optimization
- Manufacturing constraints





MotionView Introduction

This is an introductory course for solving MBD problems using HyperWorks and includes constructing, solving and post-processing multi-body dynamic (MBD) problems.

Class Format
- 3 Days
- Instructor led
- Presentation and hands-on exercises
- One student to a computer

Prerequisites
None

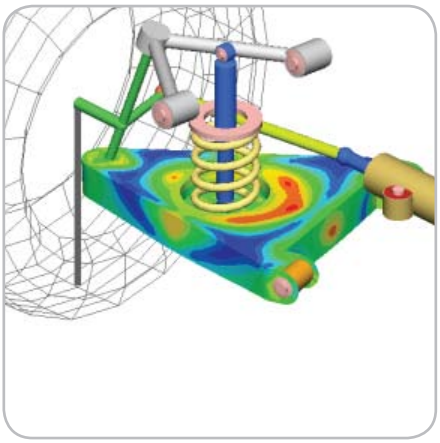
Topics
- Become familiar with the Altair HyperWorks graphical user interface Pre-processing and Solving
- Interactive model building
- CAD Import and Contact Simulation
- Flexible Body Generation and Flexible Body Dynamics Simulation
- Advanced Flexible reduction techniques
- Access Altair® HyperStudy® from MotionView to do optimization and design of experiments on an MBD model
- Automatic model building using assembly and task wizards
- MBD Model editing and checking
- Perform linear analysis on MBD models using MotionSolve Linear
- Submit XML input files to MotionSolve and adjust solver parameters
- Model setup for various solvers such as ADAMS and ABAQUS Post-processing
- Animate transient and modal results
- Animate Rigid and Flexible Body Simulation Results
- Build and edit curves
- Create and use pre-defined plot macros
- Create report templates to automate post-processing
- Create result reports with HTML, Hyper3D, JPEG, and AVI files
- Visualize and share CAE results over the Internet using the freely available Altair HyperView Player.

Suggested Follow-up Courses
MotionView MDL

MotionView MDL

Model Definition Language for MotionView

This is an advanced course for learning various aspects of MotionView's Model Definition Language (MDL). Students will learn how to construct parameterized multi-body dynamic (MBD) models and automate MBD model creation, analysis and post-processing.



Class Format

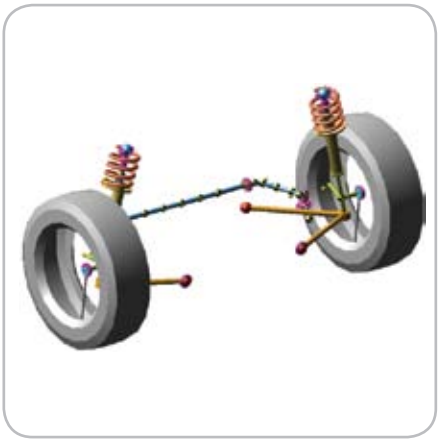
- 1 day
- Instructor led
- Presentation and hands-on exercises
- One student to a computer

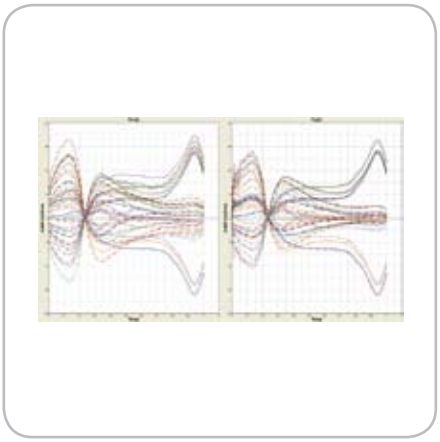
Prerequisites

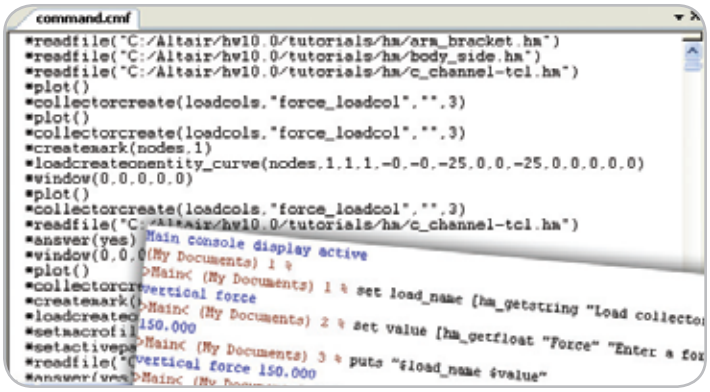
A basic knowledge of MotionView. It is recommended students complete the Introduction to MBD Problem Solving using Altair HyperWorks course.

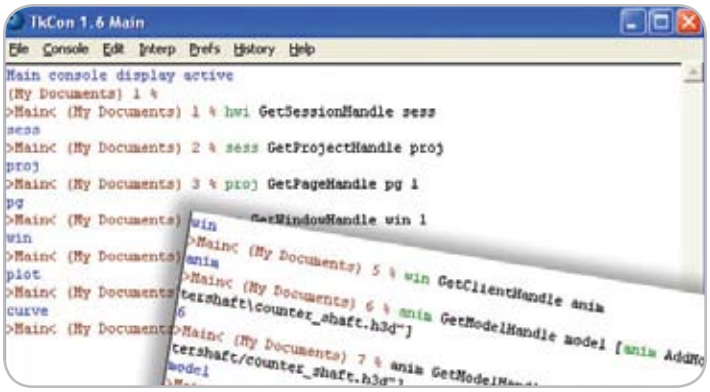
Topics

- Construct models using built-in entities (points, bodies, joints, etc.)
- Construct models using user-defined entities
- Create reusable system definitions
- Create reusable analysis-task definitions
- Create reusable datasets
- File modularization using the *Include statement
- Adding multiple instances of a system to one model
- Automatic post-processing using report definitions
- Multiple sequential simulation using command sets
- Work with various solvers using Templex templates









HyperMesh Customization Introduction

One of the most often requested advanced topics in HyperWorks is a class on the scripting layer that is incorporated in the HyperWorks Suite. This class has been developed for someone new to Tcl but familiar with programming.

The class is fast paced, hands-on, and focused on the basic tools to get you up and coding quickly.

Class Format
- 1 days
- Instructor Led
- Presentation and hands-on exercises
- One student to a computer

Prerequisites
HyperMesh Introduction

Topics
- TCL Structures
- HyperMesh Macros
- Using TCL to Control the HyperMesh Session
- Interacting with the HyperMesh Entities
- Retrieving Data from HyperMesh Entities
- Interaction with HyperMesh Solver Templates
- Access and Update a Card Image

Introduction to HyperWorks Desktop Customization

One of the most often requested advanced topics in HyperWorks is a class on the scripting layer that is incorporated in the HyperWorks Desktop Suite. This class covers the very basics in working with the Animation, MotionView, and Plotting Clients. This class has been developed for someone new to Tcl but familiar with programming. The class is fast paced, hands-on, and focused on the basic tools to get you up and coding quickly.

Class Format
- 1 days
- Instructor Led
- Presentation and hands-on exercises
- One student to a computer

Prerequisites
HyperMesh Introduction

Topics
- TCL Structures
- The HyperWorks Desktop Environment
- The Command Layer Explored
- Manipulating the Animation Client
- Applying Results and Querying Results in the Animation Client
- Introduction to the MotionView Model Client
- Manipulating the Plotting Client with a basic introduction to Templex

Fundamentals of Sheetmetal Formability

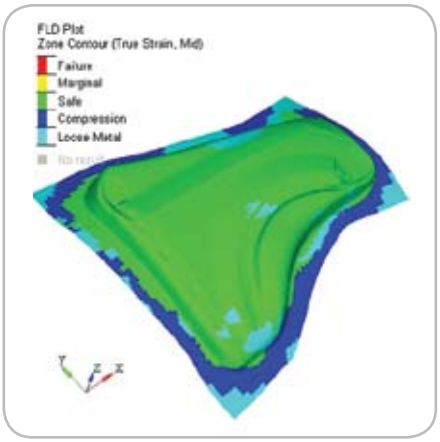
The goal of this 2-day course is to develop an Understanding of the fundamental aspects of Sheetmetal stamping. It will provide insight into the manufacturing and processing of different grades of stamping quality sheetmetal, characterization of sheetmetal behavior during stamping, methods for evaluating forming defects, the basic principles of stamping tool design, including the stamping press actions, and the state of the art computer based approach to stamping design and analysis.

Class Format

- 2 day
- Instructor led
- Presentation and hands-on exercises
- One student to a computer

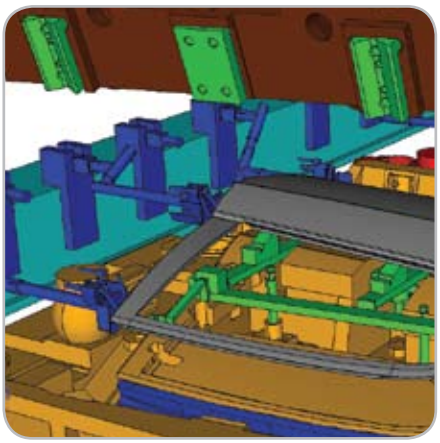
Prerequisites

None



Topics

- Manufacturing and Processing of SheetMetal
- Determination of Mechanical Properties of Sheetmetal
- Basic Deformation Modes in Stamping
- Defects in Stamping
- Test of Formability
- Basics of Draw Die Development
- Stamping Press Actions
- Advanced Forming Processes
- Computer Analysis of Forming Processes



HyperForm Introduction

1-Step and Incremental Stamping Simulation

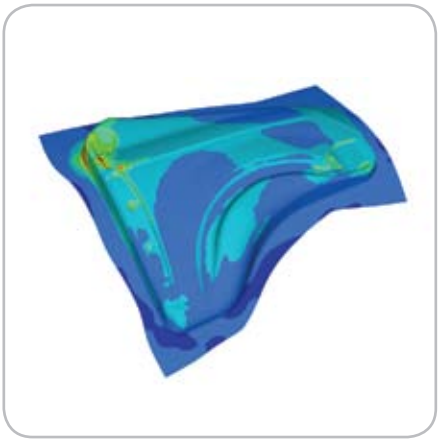
Altair HyperForm is a comprehensive FE-based sheet metal forming simulation framework. HyperForm leverages years of Altair's experience in manufacturing simulation and process design to help develop the optimal manufacturing process.

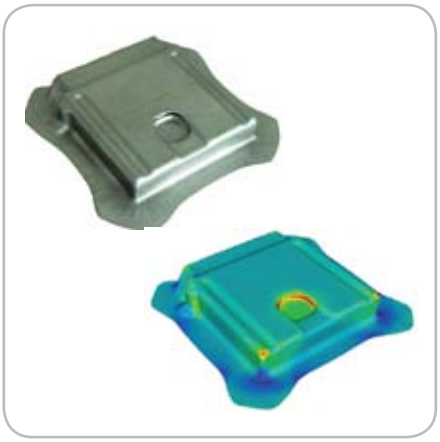
The unique HyperForm environment includes:
- Accurate blank-shape prediction and intuitive nesting interface.
- The fastest inverse solver for quick one-step analysis
- Parametric and NURBS Surface based die face development tool
- Comprehensive interface to perform incremental analysis with auto-setup and multi-stage manager for hands-off job execution

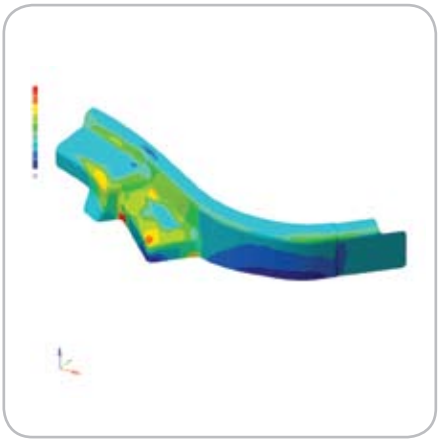
Class Format
- 3 day
- Instructor led
- Presentation and hands-on exercises
- One student to a computer

Prerequisites
None

Topics
- General Introduction
- Geometry cleanup
- Automeshing
- Mesh Quality
- Model Preparation
- 1-Step stamping simulation Increasing Blank Holder Pressure
- Applying Draw Beads and Performing Circle Grid Analysis
- Transferring Forming Results into Crash
- Laser Weld
- Trim Line Layout
- Die Module-Basic Addendum Creation
- Die Module-Designing a Parametric Addendum
- Modifying a Parametric Addendum
- Parameterization of external binder and addendum sections using section editor
- Introduction to Incremental Simulation
- Auto Process
- Simple draw forming – Manual Setup
- Combined Binder wrap and Draw Forming Analysis – Manual Setup
- Draw Beads – Manual Setup
- Spring Back – Manual Setup
- Trimming – Manual Setup
- Gravity – Manual Setup
- Redraw– Manual Setup
- Multi-Stage Manager
- Tube bending
- Hydro forming
- Blank Optimizer



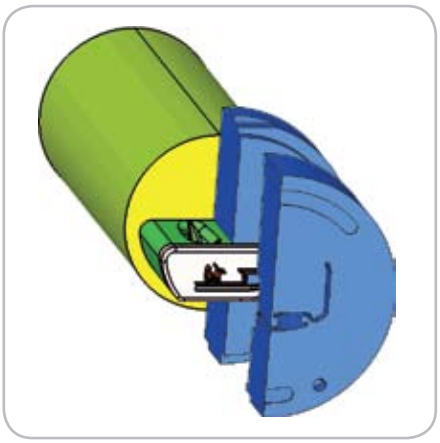




Altair® HyperXtrude®

Metal Extrusion

Today's extrusion companies are forced to produce a higher mix of complex profiles, in shorter product development cycles, while simultaneously reduce their production costs. Altair® HyperXtrude® is a simulation tool developed to analyze and validate the design of extrusion dies and process.

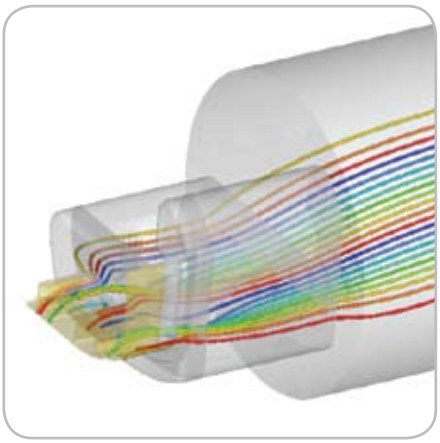


Class Format

- 3 day
- Instructor led
- Presentation and hands-on exercises
- One student to a computer

Prerequisites

Knowledge of Extrusion Process

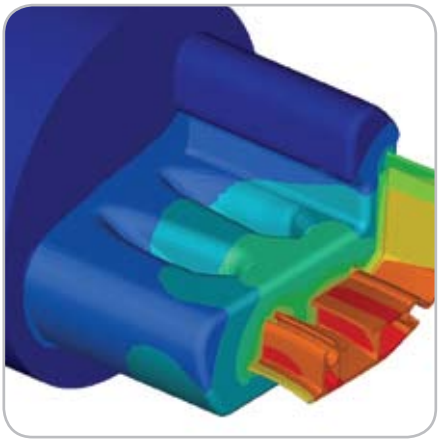


Who Should Attend

- Die Designers/Correctors
- Extrusion Engineers
- Q/C Engineers
- Metallurgists
- Tooling Engineer

Topics

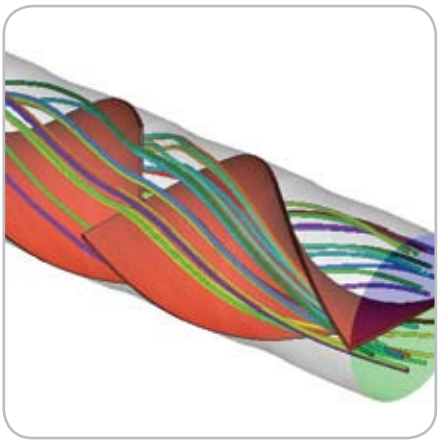
- Introduction to Extrusion
- Import CAD geometry
- Geometry cleanup
- Model preparation
- Modeling Solid and Hollow profiles
- Direct and Indirect extrusion
- Steady state and transient analysis
- Profile shape predictions
- Interpretation of results
- Die correction by adjusting bearing lengths
- Die correction by changing pocket/porthole dimensions.
- Die deflection and die stress analysis
- Material properties database
- Transverse weld length calculations
- Billet skin tracking
- Bearing optimization
- Grain size and tensile strength calculations
- User subroutines for material properties and process conditions
- User subroutines for post-analysis calculations
- Report generation



Altair® HyperXtrude®

Polymer Extrusion

Today's extrusion companies are forced to produce a higher mix of complex profiles, in shorter product development cycles, while simultaneously reduce their production costs. Altair® HyperXtrude® is a simulation tool developed to analyze and validate the design of extrusion dies and process.

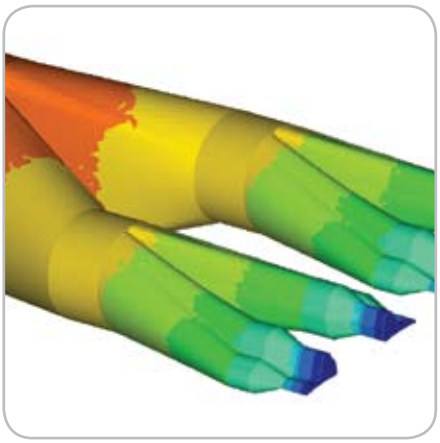


Class Format

- 2 day
- Instructor led
- Presentation and hands-on exercises
- One student to a computer

Prerequisites

Knowledge of Extrusion Process

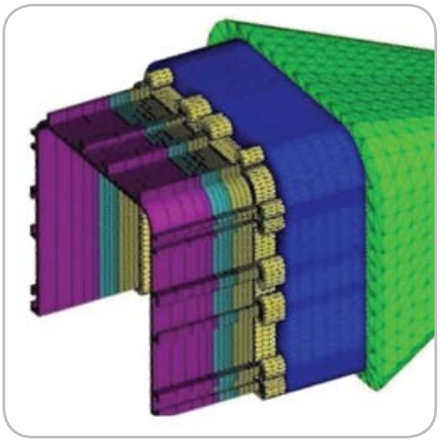


Who Should Attend

- Die Designers/Correctors
- Extrusion Engineers
- Q/C Engineers
- Tooling Engineers

Topics

- Introduction to Extrusion
- Import CAD geometry
- Geometry cleanup
- Model preparation
- Modeling Film, Sheet, and Profile dies
- Co-extrusion
- Steady state and transient analysis
- Profile shape predictions
- Interpretation of results
- Die correction by changing geometry.
- Die deflection and die stress analysis
- Polymer properties database
- User subroutines for material properties and process conditions
- User subroutines for post-analysis calculations
- Report generation



WORLD HEADQUARTERS
UNITED STATES
www.altair.com

AUSTRALIA
www.altair-anz.com

BRAZIL
www.altairengineering.com.br

CANADA
www.altairengineering.ca

CHINA
www.altair.com.cn

FRANCE
www.altairengineering.fr

GERMANY
www.altair.de

INDIA
www.altair-india.in

ITALY
www.altairengineering.it

JAPAN
www.altairjp.co.jp

KOREA
www.altair.co.kr

MEXICO
www.altairengineering.com.mx

SWEDEN
www.altair.se

UNITED KINGDOM
www.altairengineering.co.uk

Altair Engineering, Inc., World Headquarters: 1820 E. Big Beaver Rd., Troy, MI 48083-2031 USA
Phone: +1.248.614.2400 • Fax: +1.248.614.2411 • www.altair.com • info@altair.com