Theories

- What is CAD/CAM/CAE?
- Abaqus window introduction
- Introduction to different modules in Abaqus
- Units in Abaqus

CAEassistant.com

Lesson 1

Workshop Tips

- One simple simulation from A-Z step by step
- How to define isotropic elastic material
- How to use Truss element and related tips
- Using SI units
- How to apply concentrated Force
- How to define static analysis and time concept in this step
- Visualize the primary variables like stress, strain and etc.
- Get report for stress, strain, reaction forces and deflections
- How to scale results
- How to see node and Element numbers
- How to see undeformed and deformed results together

Theory

Examples

Appendix A

Appendix B

Preview

CAEassistant.com

Theories

- What is FEM?
- Familiarity with Discretization Concept
- Analytical solutions vs FEM
- How a FEM analysis is done?
- How to discretize the domain?
- How to calculate nodal solution?
- Where FEM is used
- General Usage of FEM
- An example of structural analysis
- Degree of freedom
- Convergence Study(Validation)
- Plane stress and plane strain conditions

CAEassistant.com

Workshop Tips

- Tips to start all simulations
- Create points and models in 3D space
- What is model tree and how to use it?
- How to use Probe value option to watch some results?
- How to simulate with plane stress element and usage conditions?
- What are simplify conditions to model one part of model ?
- How to apply uniform tension?
- Using remeshing rule option to check convergency?
- How to get result in specific path?

CAEassistant.com

Theories

- Introduction to ABAQUS Element Library
- Characterizing elements
- Introduction to different Types of Elements
- Different Types of degree of freedoms
- Element and node numbering
- Element formulation
- Integration Points
- Beam Element and usage conditions
- Types of Beams: Euler-Bernoulli
- Types of Beams: Timoshenko
- Beam element library of Abaqus

CAEassistant.com

Workshop Tips

- Simulation with beam Element
- How to specify beam orientation to watch some results
- How create profile for beam section
- How to see parts in different colors
- How to plot moment and force with in beam model
- Settings for plot(Font/Color/multiple plot/Legend ant etc.)
- How to model beam with 3D element
- How to define Dynamic Analysis
- How to get html report from simulation steps

CAEassistant.com

Theories

- Monitor difference in Implicit and Explicit
- Points to Choose between Explicit & Implicit
- Element library/Analysis procedures/Material models/ Contact formulation/Solution technique/ Disk space & memory in Two Solvers
- Quasi static vs static vs dynamic
- Story for Quasi static, static ,and dynamic
- Mesh refinement for two solvers
- How Abaqus solve truss problem step by step with implicit solver
- How Abaqus solve truss problem step by step with explicit solver

CAEassistant.com

Workshop Tips

- Conditions to use Axisymmetric element
- Conditions and usage of Analytical rigid part
- How to define plasticity
- How to use assembly models
- common contact definition
- How to use test results for stress-strain input and Engineering stress(strain) vs True stress(strain)
- Shell simulation and conditions usage
- Orthotropic Elastic material Definition
- How to define Functional Loading
- how to create local coordinate
- Composite layup usage
- How to see layers in details in composite materials

CAEassistant.com

Theories

- Where Explicit Analysis is used
- Stability definition for Explicit Method
- stability limit calculation
- Example for stability limit calculation
- Speeding up a Quasi static Explicit Analysis
- Load Rate Scaling
- Smooth Amplitude Curves
- Mass Scaling
- Mass VS. Load Rate Scaling
- Energy Balance

CAEassistant.com

Workshop Tips

- Using load rate scaling to increase time of computation
- Using Mass scaling to increase time of computation
- Checking proper load rate scaling/mass scaling to get right result
- Discrete rigid simulation and usage conditions
- Plot combined curve(Load vs displacement for punch)
- Plane strain simulation and condition usages
- How to visualize axisymmetric element problem to 3D
- How to visualize Plane strain element problem to 3D
- How to use display group option to remove and hide
- How to use DOF monitor
- View results at the same intervals
- View specific result in some node sets
- How to plot ALLKE), external work (ALLWK), internal energy(ALLIE), and total energy (ETOTAL)

CAEassistant.com

Theories

- Procedures Types in ABAQUS
- General and Linear Procedures
- Linear Perturbation
- Frequency Analysis
- Natural Frequencies and Mode Shapes
- Simple Example for Natural Frequencies :Mass-spring system
- Natural Freq. in Real Structures
- Eigenvalue Extraction Methods(Natural Frequencies)
- Buckling Analysis
- Stiff Structures
- Buckling of Euler column
- How FEM software solve Eigenvalue buckling problem
- Eigenvalue Extraction Method(Buckling)

CAEassistant.com

Workshop Tips

- Finding optimum time period for quasi static problem with natural frequency problem
- Viewing frequency mode shapes and natural frequency calculation
- Viewing buckling mode shapes and critical load
- Frequency analysis on a preloaded structure.
- Moment of inertia calculation
- Different ways to take a picture from viewport
- How to see node and element numbers is different modulus
- How to see thickness direction in shell model

CAEassistant.com

Theories

- Types of heat transfer analysis
- What is uncoupled heat transfer analysis and how to use it
- Heat transfer modes (convection, conduction, radiation)
- Transient vs steady state analysis
- What is Sequentially coupled thermal-stress analysis and how to use it
- What is Fully coupled thermal-stress analysis and how to use it
- What is Adaptive meshing and its advantages
- What is ALE Adaptive Meshing and its advantages
- What is Lagrangian method and its advantages
- Lagrangian Description
- What is Eulerian method and its advantages
- What is adiabatic analysis and how to use it
- Adiabatic formulas

CAEassistant.com

Workshop Tips

- Required material inputs for uncoupled steady state
- Required step settings for uncoupled steady state
- Required interaction settings for uncoupled steady state
- Required load settings for uncoupled steady state
- Required material inputs for transient thermal analysis
- Apply different types of heat modes
- Required step settings for uncoupled heat transfer
- Results of transient vs steady state
- Required material inputs for sequentially coupled analysis
- How to import result of heat transfer analysis as an input of stress analysis inputs for sequentially coupled analysis
- Settings of modulus in stress analysis
- How to create rigid spline geometry
- Usage of axisymmetric element in forge analysis
- Definition of strain-rate and temperature dependent material

CAEassistant.com

Workshop Tips

- Required material inputs for fully coupled analysis for deformable parts
- Required material inputs for fully coupled analysis for rigid parts
- Required interaction settings for fully coupled analysis
- What is thermal conductance and how to apply it
- Required inputs in edit attribute for fully coupled analysis
- Required outputs in step module for fully coupled analysis
- Abaqus Settings for ALE Adaptive meshing
- How to show Axisymmetric model as a full model
- What is Johnson-Cook plasticity and its formula
- How to define Johnson-Cook plasticity
- How to calculate equivalent force during impact
- Plot velocity vs time of impactor
- Effect of time period in impact analysis

CAEassistant.com

Theories

- What is the definition of damage and its applications?
- What is the difference between microscopic and macroscopic damage in composite materials?
- What is the various theories to simulate initiation damage in composite material structures?
- What is the available theory to simulate progressive damage of composite material structures?
- What is the elements limitations to simulate progressive damage of composite material structures?
- What is Fully coupled thermal-stress analysis and how to use it
- Which ways are available to add criterion to simulate composite damage?
- How to use composite damage results as an initiation condition of other simulation?

CAEassistant.com

Workshop Tips

- Defining Elastic property by engineering constant selection
- Defining failure criterion parameters for HASHIN
- Setting in field output to get required results
- View results in layers for fiber and matrix separately
- Recommended package to learn Composite damage simulation deeply

CAEassistant.com