

## Introduction to ABAQUS

- Linear and nonlinear structural analysis
- Static, dynamic, and heat transfer analysis
- Material models: linear elasticity and metal plasticity.
- Loads and constraints
- Modeling contact
- Selecting the appropriate elements for your problem
- Feature-based modeling, parts, and assemblies
- Working with CAD geometry and imported meshes
- Mesh generation techniques
- Creating, submitting, and monitoring analysis jobs
- Viewing simulation results

## Introduction to ABAQUS/CAE

- Creating parts using the feature-based modeler
- Importing parts into ABAQUS/CAE
- Assembling the parts using constraints
- Partitioning parts
- Meshing
- Defining analysis attributes
- Submitting and managing ABAQUS simulations
- Viewing the results of the simulations

## Introduction to ABAQUS/Standard and ABAQUS/Explicit

- Fundamental modeling techniques and input syntax
- Linear and nonlinear statics
- Selection of the appropriate element for your problem
- Adaptive load incrementation and convergence criteria
- Interpretation of messages issued by ABAQUS
- Geometric, material, and contact-induced nonlinearity
- Linear elasticity and metal plasticity
- Appropriate modeling for contact problems
- Frequency Analysis
- Buckling Analysis
- Linear and nonlinear dynamics
- Model transfer between ABAQUS/Explicit and ABAQUS/Standard