ABAQUS TRAINING

- I. Overview of ABAQUS
- II. Working with Geometry in ABAQUS
- III. Working with models Created Outside ABAQUS
- IV. Material and Section Properties
- V. Assemblies in ABAQUS
- VI. Steps, Output, Loads and Boundary Conditions
- VII. Meshing Native Geometry and Exporting Mesh
- VIII. Meshing Techniques
- IX. Constraints and Interactions
- X. Job Management and Results Visualization
- XI. Stress Concentration Factors (SCF)
- XII. Common Error Messages

CONTENTS

Session 1

- I. Overview of ABAQUS
- **II.** Working with Geometry in ABAQUS
- III. Working with models Created Outside ABAQUS

CHAPTER

I. Overview of ABAQUS

Introduction to ABAQUS 3

What is **ABAQUS**?

The heart of ABAQUS are the analysis modules, ABAQUS/Standard and ABAQUS/Explicit, which are complementary and integrated analysis tools

- <u>ABAQUS/Standard</u> is a general purpose, finite element module.
 - It provides a large number of capabilities for analyzing many different types of problems, including many nonstructural applications.
- ABAQUS/Explicit is an explicit dynamic finite element module.
- <u>ABAQUS/CAE</u> incorporates the analysis modules into a <u>C</u>omplete <u>A</u>BAQUS
 <u>E</u>nvironment for modeling, managing and monitoring ABAQUS analyses and visualizing results.

Comparing ABAQUS/Standard and ABAQUS/Explicit

- A general-purpose finite element program
 - Nonlinear problems require iterations
- Can solve for true static equilibrium in structural simulations.
- Provides a large number of capabilities for analyzing many different types of problems.
 - Nonstrctural applications
 - Coupled or uncoupled response

- A general-purpose finite element program for explicit dynamics.
 - Solution procedure does not require iteration.
- Solves highly discontinuous highspeed dynamic problems efficiently.
- Coupled-field analyses include:
 - Thermal-mechanical
 - Structural-acoustic

Modern graphical user interface of menus, icons, and dialog boxes

• Menus provide access to all capabilities

+ Abaqus/CAE 6.10-1 [Viewport: 1]	
Eile Model Viewport View Part Shape Feature Tools Plug-ins Help ₹?	-
: 🗅 🚰 🖩 🖶 🕁 🍼 🔍 🔀 🄃 🛔 🗄 🌔 ! 📐 💷 🛛 🖳 🐚 🎞 🎰	P 😭 i i 🖉
1 2 3 4 Å 0 0 0 0 4 Part defaults	
Module: Part Model: Model: Model:	
	<u> </u>
	D:

 Dialog boxes allow you to input alphanumeric information and to select various options

Young's	Poisson's
Modulus	Ratio
1	

 Icons accelerate access to frequently used features



Consistent environment

- Functionality is presented in modules
- Each module contains a logical subset of the overall functionality
- Once you understand the presentation of one module, you can easily understand the presentation of the other modules.



Part	Property	Assembly
-Create the	- Define materials	- Position parts for initial
part geometry	-Define and assign	configuration. Assembly is created for models with only one part
	sections to parts	
Step	Interaction	Load
- Define analysis steps and output	- Define contact and other	-Apply loads,
requests	steps in the analysis history	BCs and
		assign them
		to steps in the
		analysis history
Mesh	Job	Visualization
-Split assembly	- Submit, manage and monitor	- Examine
into meshable	analysis job.	results
regions and		
mesh		

Model Tree

- The Model Tree provides you with a graphical overview of your model and the objects that it contains
- It provides a convenient, centralized tool for moving between modules and for managing objects
- Some features of the model tree are discussed next



Feature-based and parametric

- A feature is a meaningful piece of the design. Models are constructed from numerous features; for example
- Geometric features
 - Solid extrusion, wire, cut, fillet, etc.
- Mesh features
 - Partition the mesh into different regions for different meshing techniques, seed different edges with different mesh densities, etc.



Part with several annotated features

- A parameter is a modifiable quantity that provides additional information for a feature; for example:
- Solid extrusion parameters
 - Sketch of extrusion cross-section, depth of extrusion.
- Cut
 - Sketch of cut cross-section, depth of cut.
- Fillet
 - Fillet radius.





- Features often have parent-child relationships, such that the existence of the child depends on the existence of the parent; for example:
 - Delete the solid extrusion, and the hole cannot exist.
 - Delete the solid extrusion, and the fillet cannot exist.
 - Delete the part, and the mesh cannot exist.
- ABAQUS/CAE always asks to make sure that you want to delete the parents and its child features.



- Parent: solid extrusion
- Child: cut
- Child: fillet

Example of Parent-Child Relationships among Features

The model database (file extension .cae)

• Only one model database can be opened in ABAQUS/CAE at a time

Replay (.rpy) file

• All commands executed during a session including any mistakes, are saved in this file

Journal (.jnl) file

• All commands necessary to recreate the most currently saved model database (.cae) are saved in this file.

Recover (.rec) file

• All commands necessary to recreate the model database (.cae) since it was most recently saved are saved in this file.

Input (.inp) file

• ABAQUS input files

Output (.odb) file

• ABAQUS output files

ABAQUS Conventions

UNITS

- ABAQUS uses no inherit sets of units
- It is the user's responsibility to use consistent units

COORDINATE SYSTEMS

- For boundary conditions and point loads, the default coordinate system is the rectangular Cartesian system.
 - Alternative local rectangular, cylindrical, and spherical systems can be defined.
 - These local directions do not rotate with the material in large- displacement analyses.

MATERIAL DIRECTIONS

- The default coordinate systems depends on the element type:
 - Solid elements use global rectangular Cartesian system.
 - Shell and membrane elements use a projection of the global rectangular Cartesian system onto the surface.



membrane elements

CHAPTER

II. Working with Geometry in ABAQUS

PARTS

Geometry built directly in ABAQUS/CAE

- Geometry built using the tools available in the Part module.
 - Called native geometry
 - A part created using the Part Module tools has a feature-based representation

Geometry imported from CAD systems

- Native CAD geometry
- Neutral formats
 - Imported geometry does not have a feature-based representation



PARTS

Parts can be scaled and/or mirrored while copying

This is available for both geometry and orphan meshes

🗖 Part Copy 🛛 🔀
Copy WHTC-SECT B to:
I
Copy Options
Compress features (geometry parts only)
Scale part by
Mirror part about X-Y plane 🔍
Separate disconnected regions into parts
OK Cancel





Part

Shape

Manager...

Create....

Rename

Copy

F

- Select the appropriate dimensionality and part type----
 - Can be changed later.
- Select the type of base feature—cannot be changed once selected.
 - The first feature you create while building a part is called the base feature.







'Datum' geometry

- A datum is a reference geometry or a construction aid that helps you create a feature when the part does not contain the necessary geometry.
- A datum is a feature of a part and is regenerated along with the rest of the part.
- A datum can be turned on and off for viewing.
- Types of datum features: point, axis, coordinate system, plane

🗖 Create Datum 🛛 🛛 🔀	🗖 Create Datum 🛛 🛛 🔀
Type Point Axis Plane CSYS Method Enter coordinates Offset from point	Type Point Axis Plane CSYS Method Principal axis Intersection of 2 planes
Midway between 2 points Offset from 2 edges Enter parameter Project point on face/plane Project point on edge/datum axis	Straight edge 2 points Axis of cylinder Normal to plane, thru point Parallel to line, thru point 3 points on circle Rotate from line

🛛 Create Datum 🛛 🛛 🔀	<u>T</u> ools	Plug-ins	Help
Type Point Axis Plane CSYS Method 3 points Offset from CSYS 2 lines	Query <u>R</u> eference Point Attachment <u>S</u> et Surface		
🗖 Create Datum 🛛 🔀	Datum		N
Type Point Axis Plane CSYS Method Offset from principal plane Offset from plane 3 points Line and point Point and normal Midway between 2 points Rotate from plane	Geo Mid CAI Dişi Vie Cu: <u>O</u> pl	ometry <u>E</u> dit Isurface D Paramete play Group <u>w</u> Cut stomize tions	• •

Partitioning in the Part module

- Partitioning subdivides the part into different regions. Each region must be assigned material and cross-sectional properties.
- Every "instance" of the part in the assembly has the same partitions.
- Regions can be used for creating geometry sets and for meshing.
 - Geometry sets created in the Part module are also available in the assembly-level modules.
 - Partitioning to create meshable regions often is better done in the Assembly and Mesh modules

Invoking the Sketcher

- When you create a base feature, ABAQUS/CAE automatically invokes the Sketcher with a blank sheet of virtual graph paper.
- An alternative is to create a "stand-alone" sketch using the Sketch module. Such sketches are not initially associated with a part but can be used later





Trimming, extending, breaking, and merging edges

- Trim/extend edges
 - Breaks the edges at intersection
- Auto-trim
 - Based on pre-selection
 - Does not break the intersecting edges
- Break edges
 - Creates separate pieces with a common vertex
- Merge edges
 - Close small gaps in a sketch





Dimensions in the Sketcher

- After creating sketch geometry, you can add dimensions between geometric entities in the sketch
- You can refine the sketch by modifying the dimension values. The sketch actively changes to reflect the new dimension values
- Dimensions serve as comments for future reference.



Adding features to a base feature

- When adding sketched features to a base feature, select the sketching plane from a planar face on the part or a datum plane.
- You can control the sketch orientation by selecting a planar face and an edge.
- By default, the edge will appear vertical and on the right side of the sketch.









Geometry Repair

Repair options

- Tools are available for manually repairing geometry:
 - Repair part by merging small edges and surfaces to make it valid
 - Replace or remove faces
 - Create cells using existing faces
 - Stitch small gaps
 - Etc.





CONTENTS

III. Working with models Created Outside ABAQUS

- An existing mesh can be imported from an ABAQUS input (.inp) or output database (.odb) file
- The mesh is called an orphan mesh because it has no associated parent geometry





b By default the imported mesh is considered a single part

• The **Part Copy** tool, however, can be used to separate disconnected regions of the model into individual parts.

🗖 Part Copy 🛛 🔀	🗖 Part Copy 🛛 🔀		
Copy Part-3-mesh-1 to:	Copy Part-1-mesh-1-Copy to:		
Part-3-mesh-1-Copy	Part-1-mesh-1-Copy-Copy		
Copy Options	Copy Options		
Compress features (geometry parts only)	Compress features (geometry parts only)		
Scale part by	Scale part by		
🗌 Mirror part about X-Y plane 🛛 🖂	Mirror part about X-Y plane 💌		
Separate disconnected regions into parts	Separate disconnected regions into parts		
OK Cancel	OK Cancel		

Orphan mesh editing

- Even though an orphan mesh has no associated geometry, modifications can be made to the mesh within the **Mesh** module.
- A suite of mesh editing tools is provided:
 - Create nodes or elements
 - Change one or more coordinates of a set of nodes
 - Verify and flip shell element normals
 - Convert elements from first-order to second –order and vice versa
 - Re-mesh a planar, triangular orphan mesh

🔲 Edit Mesh		🔲 Edit Mesh	
Category Node Element Mesh Refinement	Method Create Delete Flip normal Orient stack direction Collapse edge (tri/quad) Split edge (tri/quad) Swap diagonal (tri) Split (quad to tri) Combine (tri to quad) Renumber	Category Node Element Mesh Refinement	Method Create Edit Project Delete Merge Adjust midside Renumber
Undo		Undo	
l') Undo	Setti	🗂 Undo	Settings

M <u>e</u> sh	<u>A</u> daptivity	Feat <u>u</u> re	<u>T</u> ools	
⊆or	<u>C</u> ontrols			
Elei	Element <u>T</u> ype			
Glo	bal <u>N</u> umbering	Control		
<u>P</u> ar	t			
<u>R</u> e(<u>R</u> egion			
<u>D</u> el	Delete Part Mesh			
Delete Region Mesh				
Create <u>B</u> ottom-Up Mesh				
Associate Mesh with Geometry				
Edi	t	N		
Cre	eate Mesh P <u>a</u> rt	VS		
<u>V</u> erify				

Re-meshing of planar orphan meshes

- Mesh consists of either first or second order triangular elements; re-meshed based on boundary sizes or new uniform global size.
- Overall material area is preserved.



Orphan Mesh Export

- A native mesh created in ABAQUS mesh module can be exported by creating a mesh part.
- The mesh part is also called an orphan mesh because it has no associated parent geometry
- When creating a mesh part, all the pre-defined sets will not be available. New sets need to be defined for the newly created mesh part.





CONTENTS

Session 2

- I. Material and Section Properties
- II. Assemblies in ABAQUS
- III. Steps, Output, Loads and Boundary Conditions

CHAPTER

I. Material and Section Properties
ABAQUS Material Models

- ABAQUS has an extensive material library that can be used to model most engineering materials, including:
 - Metals
 - Plastics
 - Rubbers
 - Foams
 - Composites
 - Concrete
 - Geo-materials

In this session we discuss the most commonly used material models

- Linear elasticity
- Metal Plasticity

Linear Elasticity

Most materials have some range of deformation in which their behavior remains elastic. Quite frequently, as in the case of ductile metals, the range of elastic behavior is very limited.

► A linear elastic material model:

- Is valid for small elastic strains (normally less than 5%);
- Can be isotropic, orthotropic, or fully anistropic; and
- Can have properties that depend on temperature and/or other field variables.

Orthotropic and anisotropic material definitions require the use of local material directions

Linear Elasticity

For a linear elastic material, Hooke's law states

stress α strain

The generalized form of the law is written as

 $\sigma = D^{el}$: ε^{el}

where σ is the Cauchy (or "true") stress, D^{el} is the fourth-order elasticity tensor, and ϵ^{el} is the elastic log strain



Edit Material				
Name: Material-1			Mat <u>e</u> rial	<u>ection</u>
Description:	Edit		<u>M</u> anager	·
Material Behaviors			⊆reate	• N
Elastic			Edit	¢٢
			Сору	•
		r	<u>R</u> ename	•
General Mechanical Intermal Other	Delete		<u>D</u> elete	- F
Elastic			E <u>v</u> aluate	• •
Types Isotropic	▼ Suboptions	L		
Use temperature-dependent data				
Number of hield variables: 0		Mechanical Ihermal	<u>O</u> ther	
Moduli time scale (ror viscoelasticity): Long-term	Isotropic	Plasticity		<u>Elastic</u> Hyperelastic
	Engineering Constants	Damage for D <u>u</u> ctile	Metals	 Hyperfoam
	Lamina	Damage for Tractic	n Separation Laws	Low Density F
Young's Poisson's	Anisotropic	Damage for Figer-F	Reinforced Composites mers	 Hypoelastic Porous Elastic
Modulus Ratio	Traction	Deformation Plastic	ity	Viscoelastic
1	Shear	<u>D</u> amping		_
		E <u>x</u> pansion Prittle Creating		
	\mathbb{R}	Eos		
		Viscosity		
	Cancel			

ΛČ

Metal Plasticity

Typical uniaxial stress-strain data for an elastic-plastic solid metal are shown below:



- When most metals are loaded below their yield point, their behavior is approximately linear and elastic.
- If the stress in the metal is greater than the yield stress, most metals begin to deform plastically

Metal Plasticity

- In most metals the yield stress is a small fraction-typically 1/10% to 1% of the elastic modulus. Thus, the elastic strain in the metal is never more than this same fraction: 1/10% to 1%.
 - Consequently, the elastic response of the metal can be modeled as linear.
 - In ABAQUS all metal plasticity models are associated with linear elasticity.

A very large change in modulus occurs at yield

If the material is strained beyond yield and the strain is then reversed, the material immediately recovers its elastic stiffness.

Metal Plasticity

Hardening

- The yield surface may change as a result of plastic deformation. The change in the yield surface is defined by the hardening law.
- The following hardening laws are available in ABAQUS:
 - Perfect plasticity
 - Isotropic hardening

Intended for applications such as crash analyses, metal forming, and general collapse studies

- Kinematic hardening
- Combined isotropic/kinematic hardening

Intended for applications involving cyclic loading

Johnson-Cook plasticity

Well suited to model high-strain rate deformation of metals; only available in ABAQUS/Explicit

🗖 Edit Material 🔀		
Name: Material-1		
Description: Edit	Mat <u>e</u> rial :	Section
	Manage	r
Elastic	Create	
Plastic		
	Edit	P 0
	Сору	- F
General Mechanical Dhermal Other Delete	. <u>R</u> ename	 ••
	Delete	►
	Evaluata	
Use strain-rate-dependent data		
Use temperature-dependent data		
Number of field variables: 0 🐡	Mechanical Ihermal Other	Delete
Data	Elasticity 🕨	
Yield Plastic	Plasticity	Plastic
1	Damage for Duccle Mecais	Cap Plasticity *%
	Damage for Fiber-Reinforced Composites 🕨	Clay Plasticity
	Damage for Elastomers	Concr <u>e</u> te Damaged Plasticity
	Deformation Plasticity	Concrete Smeared Cracking
	<u>D</u> amping Expansion	Crushable <u>F</u> oam Drucker Prager
	Brittle Cracking	Mohr Coulomb Plasticity
	Eos	Porous <u>M</u> etal Plasticity
	Viscosity	Creep
OK		<u>S</u> welling
		<u>v</u> iscous

Profile Properties

- Section properties contain additional dimensional or element-type information necessary for applying material properties to a deformable body.
 - E.g, the thickness of a shell or two-dimensional solid or the cross-sectional dimensions of a beam are considered section properties.

	🔲 Create Profile 👘 🔀	Edit Profile	×
1	Name: Profile-1	Name: Profile-1	
Profile Composil	C Shape	Shape: I	
<u>M</u> anager	Box	≜ 2 ^k I	
Create	Pipe		
Edit 🔊	Circular		
Copy 🕨	Rectangular Hexagonal	t + + 1 ^{b2:}	
Copy P	Trapezoidal	¹² h ^{t1:}	
Rename P	I		
Delete 🕨	L	t3:	
	T Arbitranu		
	Generalized	T - b,	
	Continue Cancel	OK Cancel	

Section Properties

- For beams and shells the stiffness of the section is computed numerically by integrating the section properties
 - The stiffness may be computed either before the analysis (done once, efficient for linear analysis) or during the analysis



🗖 Edit Beam Section 🛛 🔀
Name: Section-1
Type: Beam
Section integration: 💿 During analysis 🔘 Before analysis
Profile name: Profile-1 Create
Profile shape: I
Basic Stiffness Fluid Inertia
Material name: Material-1 💽 Create
Section Poisson's ratio: 0
Temperature variation:
 Linear by gradients
Interpolated from temperature points
OK Cancel

Section Properties

Section	<u>P</u> rofile	Composite	
<u>M</u> ana	ger		
⊆reat	e	N	
Edit		45	×.
Сору		•	
<u>R</u> ena	me		•
Deleb	e		•
<u>A</u> ssigi	nment Ma	nager.	

Create Section		
Name: Sectio	n-1 Type	
 Solid Shell Beam Fluid Other 	Homogeneous Composite Membrane Surface General Shell Stiffness	
Continue Cancel		

Edit Section	×
Name: Section-1 Type: Shell / Continuum Shell, Homogeneous	
Section integration: During analysis Before analysis Basic Advanced	
Thickness Shell thickness: Image: Comparison of the stribution: Image: Comparison of the stribution of the stribution: Image: Comparison of the stribution of the stributio	
Material: Material-1 Create Thickness integration rule: O Simpson O Gauss Thickness integration points: 5	
Options: Rebar Layers	
OK Cancel	

CHAPTER

II. Assemblies in ABAQUS

- An assembly contains all the geometry included in the finite element model.
- **Each ABAQUS/CAE model contains a single assembly.**
- An assembly is empty initially even if you have created some parts.
- An assembly does not contain parts directly; instead, it contains "instances" of parts.
- ► For convenience, instances can be turned on and off for viewing.
- The following points explain the relationship between parts, part instances, and assemblies.

Parts

- You create parts in the Part module
- Each part is a distinct geometric entity that can be modified and manipulated independently of other parts
- Each part exists in its own coordinate system and has no knowledge of other parts
- Each part has it owns references section properties



Example: five parts used to model a child's wagon

Part instances and assemblies

- You create instances of your parts in the Assembly module
 - An instance always maintains its association with the original part
 - You can create instance of a part many times and assemble multiple instances of the same part
 - Each instance of the part is associated with the section properties assigned to the part in the Property module
- You position part instances relative to each other in a global coordinate system to form the assembly
- You can modify the original part, and ABAQUS/CAE updates any instances of that part when you return to the Assembly module.

Dependent and independent part instances

- You can create either independent or dependent part instances.
- An independent instance is effectively a copy of the part and can be modified
- A dependent instance shares the geometry and the mesh of the original part and cannot be modified.
- Dependent instances are created by default

🗖 Create Instance 🛛 🔀		
Parts		
Part-1 Part-1-mesh-1 Part-1-mesh-1-Copy		
Instance Type Opendent (mesh on part)		
O Independent (mesh on instance)		
Note: To change a Dependent instance's mesh, you must edit its part's mesh.		
Auto-offset from other instances		
OK Apply Cancel		

Example: Assembly of a child's wagon.





One instance of the body Two instances of the axle Four instances of the axle mount One instance of the handle Four instances of the wheel





The wheel instances are automatically updated when the part definition changes.



Positioning Part instances

<u>er</u>

(XYZ)

- Positioning is the main task in the Assembly module. Two general methods
 - Absolute positioning
 - Relative positioning
- Absolute positioning is not treated as a feature of the assembly
 - Translation
 - Rotation
 - Replace one part with another



Sets and Surfaces

Sets define regions from one or more parts or part instances for:

- Loads and boundary conditions
- Generating output during the analysis
- Surfaces define regions from one or more parts or part instances for specifying:
 - Contact
 - Distributed loads
- Sets and surface are useful when certain geometric groups will be used for multiple purposes.
- Note: Often it is more convenient to select geometric entities directly using the mouse when defining loads, boundary conditions, and fields rather than using sets and surfaces. One advantage of sets is that you provide names that will be meaningful for results visualization.

Sets and Surfaces

- Sets and Surfaces can be defined at the part level or the assembly level (i.e, associated with the part instance rather than the part itself)
 - Part sets appear in the Model Tree in a Set container under the part with which they are associated.
 - Sets from an instanced part appear in the Model Tree under the assembly.
 - Part sets are also available in the assembly-related modules.
 - Only "read-only" access to these sets is provided in the assembly-related modules.



CHAPTER

III. Steps, Output, Loads and Boundary Conditions

Analysis Steps and Procedures

A basic concept in ABAQUS is the division of the problem history into steps.

- A step is any convenient phase of the history a thermal transient, a creep hold, a dynamic transient, etc.
- In its simplest form a step can be just a static analysis of a load change from one magnitude to another.

► For each step the user chooses an analysis procedure.

 This choice defines the type of analysis to be performed during the step: static stress analysis, dynamic stress analysis, eigenvalue buckling, transient heat transfer analysis, etc.

The rest of the step definition consists of load, boundary, and output request specifications.

Analysis Steps and Procedures

ABAQUS distinguishes between two kinds of analysis procedure:

General analysis procedures

- Response can be linear or nonlinear
- Steps that use general procedures are known as general steps
- The starting point for each general step is the state of the model at the end of the last general step.

Linear pertubation procedures

- Response can only be linear.
- The linear pertubation is about a base state, which can be either the initial or the current configuration of the model.
 - Response prior to reaching the base state can be nonlinear
- Steps that use linear procedures are known as pertubation steps

Analysis Steps and Procedures

Defining Steps in ABAQUS/CAE

💻 Create Step 🛛 🔀			🗖 Create Step 🛛 🔀		
		Name: Step-1		Name: Step-1	
		Insert new step after	Π	Insert new step after	
<u>S</u> tep <u>O</u> utput	:	Initial	Π	Initial	
<u>M</u> anager			Π		
<u>C</u> reate	N		Π		
Edit	N		Π		
Replace	•		Π		
<u>R</u> ename	F	Procedure type: General 🛛 👻	Π	Procedure type: Linear perturbation 🔽	
<u>D</u> elete	•	Dynamic, Explicit		Buckle	
<u>S</u> uppress	•	Dynamic, Temp-disp, Explicit Frequency		Frequency	
Res <u>u</u> me	•	Geostatic	Π	Static, Linear perturbation	
<u>N</u> lgeom		Heat transfer	Π	Steady-state dynamics, Direct	
		Soils	Π		
		Static, General	Π		
		Static, Riks 💌			
		Continue Cancel		Continue Cancel	

Output

Two types of output data: field and history data

- Field data is used for model (deformed, contour, etc.) and X-Y plots.
- History data is used for X-Y plots.





🗖 Edit Fie	eld Output Request	×
Name:	F-Output-1	
Step:	Step-1	
Procedure:	Static, General	
Domain:	Whole model	
Frequency:	Every n increments 💌 n: 1	
Timing: C	Dutput at exact times	
COutput Var	riables	
📀 Select fi	rom list below 🔘 Preselected defaults 🔵 All 🔘 Edit variables	
S,MISESMA	XX, TSHR, CTSHR, ALPHA, TRIAX, VS, PS, CS11, ALPHAN, SSAVG, U, UT, UR, V, VT, VR, R	.E
 St St Dit Fc Cc Er Fa Th Note: Err 	resses rains splacement/Velocity/Acceleration orces/Reactions ontact nergy ailure/Fracture nermal	
Output at sh	ell, beam, and lavered section points:	
() Use de		
Include la	ocal coordinate directions when available	
	OK Cancel	

Field Output

History Output



Name: H-Output-1 Step: Step-1 Procedure: Static, General Domain: Whole model Frequency: Every n increments r: 1 Timing: Output at exact times Output Variables IRA1,IRA2,IRA3,IRA1,IRAR2,IRAR3,IRF1,IRF2,IRF3,IRM1,IRM2,IRM3 IRA1,IRA2,IRA3,IRA1,IRAR2,IRAR3,IRF1,IRF2,IRF3,IRM1,IRM2,IRM3 IRA1,IRA2,IRA3,IRA1,IRAR2,IRAR3,IRF1,IRF2,IRF3,IRM1,IRM2,IRM3 IRA1,IRA2,IRA3,IRA1,IRAR2,IRAR3,IRF1,IRF2,IRF3,IRM1,IRM2,IRM3 IRA1,IRA2,IRA3,IRA1,IRAR2,IRAR3,IRF1,IRF2,IRF3,IRM1,IRM2,IRM3		
Step: Step-1 Procedure: Static, General Domain: Whole model Frequency: Every n increments n: 1 Timing: Output at exact times Output Variables Output Variables Select from list below Preselected defaults All Edit variables IRA1,IRA2,IRA3,IRAR1,IRAR2,IRAR3,IRF1,IRF2,IRF3,IRM1,IRM2,IRM3		
Procedure: Static, General Domain: Whole model Frequency: Every n increments Timing: Output at exact times Output Variables Select from list below Preselected defaults All Edit variables IRA1,IRA2,IRA3,IRAR1,IRAR2,IRAR3,IRF1,IRF2,IRF3,IRM1,IRM2,IRM3 IRA1,IRA2,IRA3,IRA1,IRAR2,IRAR3,IRF1,IRF2,IRF3,IRM1,IRM2,IRM3 P Displacement/Velocity/Acceleration P Forces/Reactions Contact D Energy Frailure/Fracture Thermal		
Domain: Whole model Frequency: Every n increments n: 1 Timing: Output at exact times Output Variables Select from list below Preselected defaults All Edit variables IRA1,IRA2,IRA3,IRA1,IRAR2,IRAR3,IRF1,IRF2,IRF3,IRM1,IRM2,IRM3 Displacement/Velocity/Acceleration PiForces/Reactions Contact Energy Failure/Fracture Thermal Output for rebar		
Frequency: Every n increments Iming: Output at exact times Output Variables Select from list below Preselected defaults All Edit variables IRA1,IRA2,IRA3,IRA1,IRAR2,IRAR3,IRF1,IRF2,IRF3,IRM1,IRM2,IRM3 IRA1,IRA2,IRA3,IRA1,IRAR2,IRAR3,IRF1,IRF2,IRF3,IRM1,IRM2,IRM3 Image: Contact Image: Conta		
Timing: Output at exact times Output Variables Select from list below Preselected defaults All Edit variables IRA1,IRA2,IRA3,IRA1,IRAR2,IRAR3,IRF1,IRF2,IRF3,IRM1,IRM2,IRM3 IRA1,IRA2,IRA3,IRA1,IRAR2,IRAR3,IRF1,IRF2,IRF3,IRM1,IRM2,IRM3 Poisplacement/Velocity/Acceleration Poisplacement/Velocity/Acceleration Porces/Reactions Output for rebar Output for rebar		
Output Variables Select from list below Preselected defaults IRA1,IRA2,IRA3,IRA1,IRAR2,IRAR3,IRF1,IRF2,IRF3,IRM1,IRM2,IRM3 Image: state s		
 Select from list below Preselected defaults All Edit variables IRA1,IRA2,IRA3,IRAR1,IRAR2,IRAR3,IRF1,IRF2,IRF3,IRM1,IRM2,IRM3 Displacement/Velocity/Acceleration Forces/Reactionsi Contact Energy Failure/Fracture Thermal 		
IRA1,IRA2,IRA3,IRAR1,IRAR2,IRAR3,IRF1,IRF2,IRF3,IRM1,IRM2,IRM3		
 Displacement/Velocity/Acceleration Forces/Reactions Contact Energy Failure/Fracture Thermal Qutput for rehar 		
 Forces/Reactions Contact Energy Failure/Fracture Thermal 		
 Contact Energy Failure/Fracture Thermal 		
Energy Failure/Fracture Thermal		
Thermal Output for rehar		
Output for rebar		
Output for rebar		
estration teen		
Output at shell, beam, and layered section points:		
⊙ Use defaults 🔿 Specify:		
OK Cancel		

Loads and Boundary Conditions

- In ABAQUS the term load (as in the Load module in ABAQUS/CAE) generally refers to anything that includes a change in the response of a structure from its initial state, including:
 - Concentrated forces,
 - Surface tractions (e.g., pressure, shear, etc.)
 - Nonzero boundary conditions,
 - Body loads, and
 - Temperature (with thermal expansion of the material defined)
- Sometimes the term refers specially to force-type quantities (as in the Load Manager of the Load module); for example,
 - Concentrated forces, pressures, and body loads but not boundary conditions or temperature.

Loads and Boundary Conditions



Loads

Defining loads and boundary conditions in ABAQUS/CAE

• Create a load (or boundary condition), and select the steps in which it will be active.



🔲 Create Load	\mathbf{X}
Name: Load-1	
Step: Step-1	¥
Procedure: Static, Ge	eneral
Category	Types for Selected Step
Mechanical	Concentrated force
◯ Thermal	Moment
O Acoustic	Pressure
() Fluid	Shell edge load
Electrical	Surface traction
	Pipe pressure
	Line load
Other	Gravity
	Bolt load 🗸 🗸 🗸
Continue	Cancel

Loads

• Select the application region (geometry, nodes, or elements selected directly using the mouse or sets and surfaces defined previously)

(Surfaces on orphan meshes can be selected by picking one element face and a feature angle. Individual "edits" make it easy to clean up anomalies.)

• Edit the load (or boundary condition) to specify magnitude, etc.



🗖 Edit Load 🛛 🛛 🔀			
Name: Loa	Load-1		
Type: Cor	Concentrated force		
Step: Ste	itep: Step-1 (Static, General)		
Region: (Picked)			
CSYS: (Global) Edit 🙏 Create			
Distribution:	Uniform Create		
CF1:			
CF2:			
CF3:			
Amplitude:	(Ramp) 🛛 Create		
Follow nodal rotation			
Note: Force will be applied per node.			
OK Cancel			

Boundary Conditions





Create Bound	ary Condition 🛛 🛛 🔀
Name: BC-1 Step: Step-1 Procedure: Static, C Category Mechanical Fluid Other	Seneral Types for Selected Step Symmetry/Antisymmetry/Encastre Displacement/Rotation Velocity/Angular velocity Connector displacement Connector velocity
Continue	Cancel



Reference Point



Session 3

- I. Meshing Native Geometry and Exporting Mesh
- **II.** Meshing Techniques
- **III.** Constraints and Interactions

CHAPTER

I. Meshing Native Geometry and Exporting Mesh

Meshing Native Geometry

General capabilities of the Mesh module

- Allows you to mesh an assembly using various levels of automation, and controls to suit the needs of your analysis
- Assign mesh attributes, and set mesh controls to specify:
 - Meshing technique
 - Element shape
 - Element type
 - Mesh density
- Generate the mesh
- Query and verify the mesh for:
 - Number of nodes and elements
 - Element type
 - Element quality
Dependent and Independent Part Instances

You can create either independent or dependent part instances

- Independent instances can be modified (e.g., they can be partitioned).
- Dependent instances cannot be modified (e.g., they cannot be partitioned)
- Dependent instances share the same geometry and mesh of the original part so any modifications must be made to the original part.
- Attributes (loads, BCs, etc.) and sets/surfaces can be created, however.

All instances of a part must be either dependent or independent

• No mixture is allowed for a given part

All orphan mesh instances must be dependent

Dependent and Independent Part Instances

- Choose Independent or Dependent when creating part instance
- Independent not allowed if:
 - Part is meshed
 - Part has Virtual Topology
 - Dependent instances of Part already exist
 - Part is an orphan mesh
- Dependent not allowed if:
 - Independent instances of Part already exist
- Can easily convert between dependent and independent



Exporting Mesh

- Allows you to export mesh of an assembly by creating a mesh part
- ► The mesh part created become an orphan mesh instances
- The mesh part is then being imported back to Assembly Module as dependent part
- ► All the sets/surfaces defined earlier have to be redefined
- Mesh part is useful when you want to duplicate the same mesh either by revolving, mirror and etc.
- These mesh part can also be exported to other FE analysis software.

Exporting Mesh

Steps in creating mesh part and exporting mesh

M <u>e</u> sh	<u>A</u> daptivity	Feat <u>u</u> re	<u>T</u> ools					
⊆or	<u>C</u> ontrols							
Elei	ment <u>T</u> ype							
Glo	bal <u>N</u> umbering	g Control						
Īns	tance							
<u>R</u> eç	jion							
<u>D</u> el	<u>D</u> elete Instance Mesh							
Delete Region Mesh								
Cre	Create <u>B</u> ottom-Up Mesh							
Associate Mesh with Geometry								
<u>E</u> dit								
Cre	Create Mesh P <u>a</u> rt							
<u>V</u> er	₩							

<u>F</u> ile	<u>M</u> odel	Vie <u>w</u> port	<u>V</u> iew	<u>S</u> eed	M <u>e</u> sh	<u>A</u> daptivity	Fe
N	ew Model	Database			•		
<u>0</u>	pen			Ctrl+	-0 💾		
N	etwork Ol	DB Connecto	or		- ▶		
⊆	lose ODB.				M	dule: Mesh	
S	et <u>W</u> ork D	irectory					
5	ave			Ctrl-	FS [. [_]	
S	ave <u>A</u> s					 H	
C	ompress N	М <u>D</u> B			1	, 1	
S	a <u>v</u> e Optio	ns					
Īī	nport				► L	₩ ₩ ₩	
E:	xport				- Þ.	<u>S</u> ketch	
<u>R</u>	un Script.					<u>P</u> art	2
M	lacro Man	ager				<u>A</u> ssembly	~
<u>P</u> I	rint			Ctrl-	+P	<u>V</u> RML	
A	<u>b</u> agus PDI	E				<u>3</u> DXML	
1	C://WF	HTC FE/WHI	C-SECT	B.cae	1	š –	
2	<://₩ŀ	HTC FE/WHI	IC-SECT	A3.cae	+		
3	C://CP	P FE/CPP - S	5ECT A3	.cae		R, 📥 📃	
4	⊂://₩ŀ	HTC FE/WHI	C-SECT	B_try.ca	ae 🚺		
E	<u>x</u> it			Ctrl+	Q	- 60	

CHAPTER

II. Meshing Techniques

Introduction to ABAQUS 77

Free meshing

- Free meshing uses no pre-established mesh patterns, making it impossible to predict a free mesh pattern before creating mesh
- Element shape options available for free meshing two-dimensional regions:

Quadrilateral (default)	Can be applied to any planar or curved surface
Quadrilateral dominated	Allows some triangular elements for transition
Triangular	can be applied to any planar or curved surface



Two-dimensional swept meshes

- All-quad meshing of swept regions
 - Planar or curved surfaces
- Quad-dominated meshing of degenerate revolved regions
 - (Degenerate regions include the axis of revolution)



Swept mesh



Degenerate revolved mesh

Swept Meshing

- Swept solid regions can be filled with:
 - Hex meshes
 - Hex-dominated meshes
 - Wedge meshes
- The swept path can be:
 - Straight line
 - Circular arc
 - Spline

sweep path: straight line extruded mesh sweep path: arc



Introduction to ABAQUS 81

Limitation

- Source and target faces must be planar
- Constant cross-sections only
- Target face and each side wall must have only one face.



Swept meshable



Not swept meshable

Structured meshing

- The structured meshing technique generates meshes using simple predefined mesh topologies
- ABAQUS transforms the mesh of a regularly shaped region, such as a square or a cube, onto the geometry of the region you want to mesh.
- Structured meshing generally gives the most control over the mesh.





three-dimensional structured meshable regions



simple mesh topology

Which regions are meshable?

- ABAQUS/CAE automatically determines meshability for each region based on its geometry and mesh controls
- Regions are color coded to indicate their currently assigned meshing technique



Partitioning to make regions meshable

- Most three-dimensional part instances require partitioning to permit hexahedral meshing
 - Complex geometries often can be partitioned into simpler, meshable regions.
- Partitioning can be used to:
 - Change and simplify the topology so that the regions can be meshed using primarily hexahedral elements with the structured or swept meshing techniques.





Mesh Compatibility

- Different regions of the same part instance can be meshed using different elements types, such as tetrahedra and hexahedra.
- Tie constraints are created automatically to connect the regions.
- Allows hexahedra to be used adjacent to contact surfaces or in high gradient regions where accuracy is essential, with tetrahedra in other regions.
- When a region is meshed, an existing mesh on an adjacent region is unaffected.

tle constraints inserted automatically at partition

Mesh Compatibility

- Currently it is not possible to automatically obtain meshes that are compatible between part instances
- If mesh compatibility is required between two or more bodies, first try to create a single part that contains all the bodies.
 - Multiple part instances can be merged into a single part instance in the Assembly module
 - Different material regions can be separated using partitions.
- If the two objects must be modeled as separate parts, consider using tie constraints to "glue" two regions together.
- Alternatively, merge instance meshes into a conforming orphan mesh.



Using tie constraints to glue the cylinder to the block: exploded view of assembly (top) and mesh

Mesh seeds

• Mesh seeds are markers that you define along the edges of a region to specify the desired, or "target," mesh density.



You can set a typical global element length for part instances.

- ABAQUS/CAE automatically creates mesh seed along all relevant edges based on the typical element length.
- New edges created by partitioning automatically inherit the global mesh seeds.

You can override the global mesh seed with local mesh seeds along selected edges

- Edge mesh seeds can be uniform or biased.
- Edge mesh seeds propagate automatically from the selected edge to the matching edges for swept meshable regions.

Global seeds (black) and local seeds (magenta)



Partitioning into different mesh regions

- Partitioning creates additional edges, which allows more control over local mesh density.
- Each mesh region can have different mesh controls.





Partitioning and local mesh seeding allows you to refine mesh in the area of a stress concentration.

Assigning Element Types

- The available element types depend on the geometry of your model
- You can assign the element type either before or after you create the mesh
- Different element types can be assigned to different regions of your model
- Items such as loads and boundary conditions depend on the uunderlying geometry, not the mesh, so performing parametric studies on mesh density or element types is very easy

M <u>e</u> sh	<u>A</u> daptivity	Feat <u>u</u> re	<u>T</u> ools			
<u>C</u> or						
Elei	ment <u>T</u> ype		N			
Glo	bal <u>N</u> umbering	g Control	K			
<u>P</u> ar	·t					
<u>R</u> egion						
<u>D</u> elete Part Mesh						
Delete Region Mesh						
Create <u>B</u> ottom-Up Mesh						
A <u>s</u> sociate Mesh with Geometry						
<u> </u>						
Cre	Create Mesh P <u>a</u> rt					
<u>V</u> erify						

Assigning Element Types

Element Type		
Element Library Standard Explicit Geometric Order Linear Quadratic	mily oustic upled Temperature-Displacement sket at Transfer	<
Quad Tri		
Element Controls Membrane strains: Membrane hourglass stiffness:	Finite Small Specify	
Bending hourglass stiffness: Drilling hourglass scaling factor	Use default O Specify r: O Use default O Specify	~
S4R: A 4-node doubly curved the select an element shape a select an element shape a select "Mesh->Controls" from	hin or thick shell, reduced integration, hourglass control, finite membrane strains. for meshing, m the main menu bar.	
ОК	Defaults Cancel	

Element in ABAQUS

Family

- A family of finite elements is the broadest category used to classify elements.
- Elements in the same family share many basic features.
- There are many variations within a family.



Element in ABAQUS

Number of nodes (interpolation)

- An element's number of nodes determines how the nodal degrees of freedom will be interpolated over the domain of the element.
- ABAQUS includes elements with both first- and second-order interpolation.



CPE4

First-order interpolation



CPE8

Second-order interpolation

Element in ABAQUS

Integration

- The stiffness and mass of an element are calculated numerically at sampling points called "integration points: within the element
- The numerical algorithm used to integrate these variables influences how an element behaves

Full Integration

 The minimum integration order required for exact integration of the strain energy for an undistorted element with linear material properties.

Reduced Integration

• The integration rule that is one order less than the full integration rule.



CHAPTER

III. Constraints and Interactions

Interaction - Constraints

What are constraints?

Constraints allow you to model kinematic relationship between points

Tie constraints

- Allow you to fuse together two regions even though the meshes created on the surfaces of the regions may be dissimilar
 - Easy mesh transitioning
- It provides a simple way to bond surfaces together permanently
- Surface-based constraint using a master-slave formulation
- The constraint prevents slave nodes from separating or sliding relative to the master surface



Constraints - Tie





MPC Constraint

Equation

Continue...

Shell-to-solid coupling Embedded region

Cancel

Edit Constraint	×
Name: Constraint-1	
Type: Tie	
Master surface: (Picked)	
Slave surface: (Picked)	
Discretization method: Analysis default	
Exclude shell element thickness	
Position Tolerance	
 Use computed default 	
O Specify distance:	
Note: Nodes on the slave surface that are considered to be outside the position tolerance will NOT be tied.	
Adjust slave surface initial position	_
Tie rotational DOFs if applicable	
Constraint Ratio	
Ose analysis default	
O Specify value	
OK Cancel	

Constraints - Coupling

	💻 Edit Constraint 🛛 🛛 🔀	
🔲 Create Constraint 🛛	Name: Constraint-1	×RP-1
Name: Constraint-1	Type: Coupling	
Туре	Control point: (Picked) 📕	
Tie	Surface: (Picked)	Y
Rigid body	Coupling type: 💿 Kinematic	
Display body	Continuum distributing	2° ~ X
Coupling MPC Constraint	Structural distributing	
Shell-to-solid coupling Couplir	Constrained degrees of freedom:	
Embedded region	V1	
Equation	✓ U2	
	🔽 U3	
Continue Cancel	UR1	Y .
		z< ``x
	Influence radius: To outermost point on the region 	
	O Specify:	
	CSYS (Global) Edit 🙏 Create	
	OK Cancel	

Session 4

- I. Job Management and Results Visualization
- **II.** Stress Concentration Factors (SCF)
- III. Common Error Messages
- **IV.** Tutorial 1 Linear Static Analysis

CHAPTER

I. Job Management and Results Visualization

Job Management







Job Management

oob management	Job <u>A</u> daptivity	<u>C</u> o-executio
🗖 Edit Job 🛛 🗙	<u>M</u> anager	
Newsy July 1	<u>C</u> reate	
Name: Job-1	E_dit ►	
Model: Model-1	Сору 🕨	e: Job
Analysis product: Unknown	<u>R</u> ename ►	
	Delete	
	Write Input P	_
Submission General Memory Parallelization Precision	Submit	
	Continue	
Eull analysis	Monitor ►	
Descurr (Funkcik)	Results ►	
	Job <u>A</u> daptivity	<u> </u>
O Restart	<u>M</u> anager	
C Run Mode	<u>⊂</u> reate…	
Host name;	E_dit 🕨 🕨	
Sackground Queue:	Сору 🕨	e: Job
(in the second se	<u>R</u> ename •	
Submit Time	Delete ►	
 Immediately 	<u>W</u> rite Input ►	
O Wait: hrs. min.	Data Check	
		_
O At:	Continue 🕨	
		Job-1
OK	Kill N	
	Export ►	

Job Management

🔲 Job-1	Monitor							
Job: Job	-1 Status: Nor	ne						
Step	Increment	Att	Severe Discon Iter	Equil Iter	Total Iter	Total Time/Freq	Step Time/LPF	Time/LPF Inc
Log E	rrors Warning:	s Outpu	t Data File	Message F	ile Status f	-ile		
_ Search `	Text							
Text to f	ind:			Matc	h case 👢 M	lext 🕜 Previo	us	
-	Kill Dismiss							

Results Visualization



Results Visualization



🔲 Field Output	
Step/Frame Step: 1, Unity Loa Frame: 6 Step/F	ad Frame
Primary Variable Output Variable List only varia	Deformed Variable Symbol Variable Status Variable
Rame CF CM E RF RM S U U UR	Description (* indicates complex) Point loads at nodes Point moments at nodes Strain components at integration points Reaction force at nodes Reaction moment at nodes Stress components at integration points Spatial displacement at nodes Rotational displacement at nodes
Invariant Mises Max. In-Plane P Min. In-Plane P Out-of-Plane P Max. Principal Mid. Principal	Principal Principal Principal
ОК	Apply Cancel

Results Visualization


CHAPTER

II. Stress Concentration Factors (SCF)

Introduction to ABAQUS 109

Stress Concentration

- Second-order elements clearly outperform first-order elements in problems with stress concentrations and are ideally suited for the analysis of (stationary) cracks
 - Both full integrated and reduced-integration element work well.
 - Reduced-integration elements tend to be somewhere more efficient results are often as good or better than full integration at lower computational cost.
- Second-order elements capture geometric features, such as curved edges, with fewer elements than first-order elements



Model with first-orcer elements—elemert faces are straight frie segments

Model with secondorder elements---element faces are quadratic curves

Stress Concentration

 First-order elements (including incompatible mode elements) are relatively poor in the study of stress concentration problems.

Element	σ _{γγ} at D (Target=100.0)				
type	Coarse mesh	Fine mesh			
CPS3 🛆	55.06	76.87			
CPS4	71.98	91.2			
CPS4I	63.45	84.37			
CPS4R	43.67	60.6			
CPS6 🛆	96.12	101.4			
CPS8	91.2	100.12			
CPS8R	92.56	97.16			

Stress Concentration

- Second-order elements give much better results
- Well-shaped, second order, reduced-integration quadrilaterals and hexahedra can provide high accuracy in stress concentration regions
 - Distorted elements reduce the accuracy in these regions



CHAPTER

III. Common Error Messages

Introduction to ABAQUS 113

Common Errors

Do not apply boundary conditions to the slave nodes of a tie constraint. This will cause the nodes to be overconstrained, resulting in errors in the analysis.

Symptoms:

 Zero pivot warnings in the JOB Diagnostics dialog box and the message (.msg) file in ABAQUS/Standard

🔲 Job-1	🗖 Job-1 Monitor										
Job: Job	Job: Job-1 Status: Aborted										
Step	Increment	Att	Severe Discon Iter	Equil Iter	Total Iter	Total Time/Freq	Step Time/LPF	Time/LPF Inc			
Log	Log ! Errors ! Warnings Output Data File Message File Status File										
Section not poss Analysis	Section definitions are missing or incorrect for the elements indicated above. Further processing of the input file is not possible until this input error is fixed. Analysis Input File Processor exited with an error.										
Search Text Text to find: Match case Next Yerevious											
	Kill Dismiss										