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

What is ABAQUS?

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

- ABAQUS/Standard 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.

- ABAQUS/CAE incorporates the analysis modules into a **C**omplete **A**BAQUS **E**nvironment for modeling, managing and monitoring ABAQUS analyses and visualizing results.

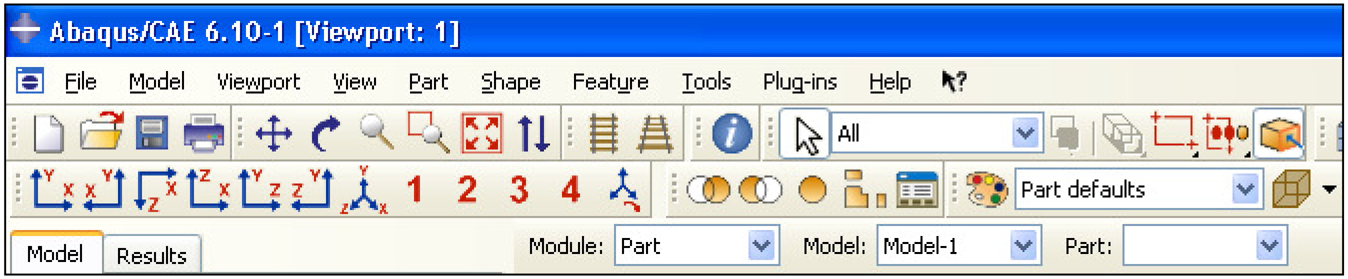
Comparing ABAQUS/Standard and ABAQUS/Explicit

- 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

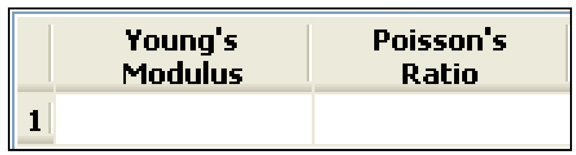
- A general-purpose finite element program for explicit dynamics.
  - Solution procedure does not require iteration.
- Solves highly discontinuous high-speed dynamic problems efficiently.
- Coupled-field analyses include:
  - Thermal-mechanical
  - Structural-acoustic

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

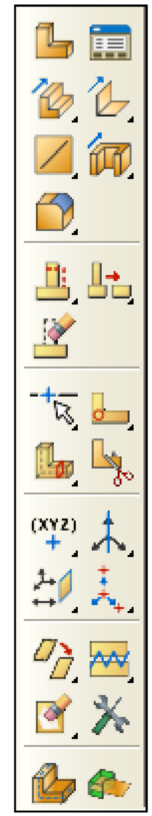
- Menus provide access to all capabilities



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

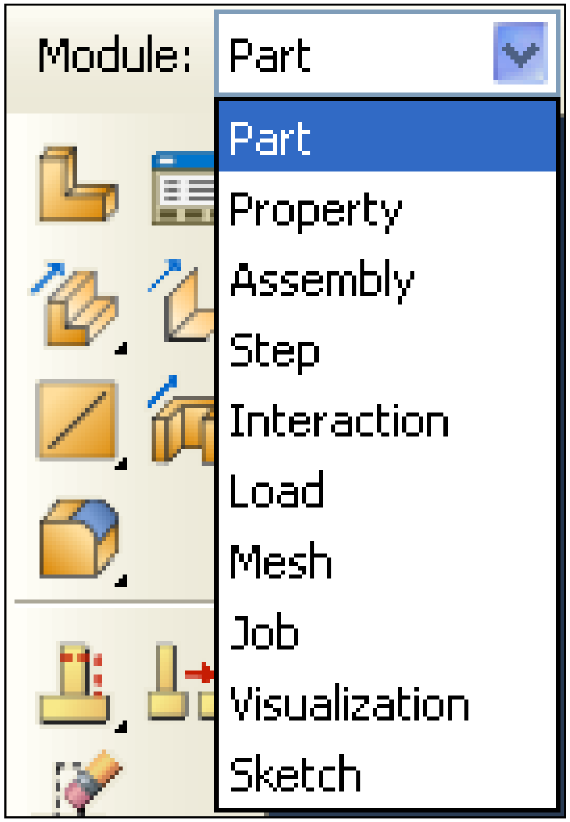


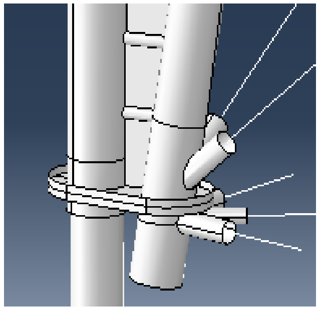
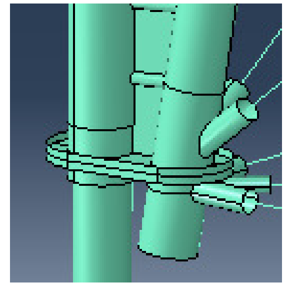
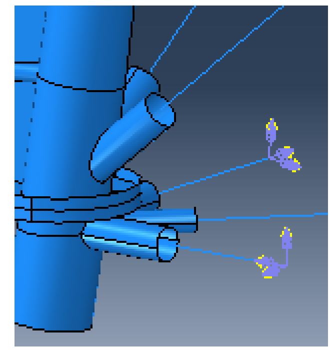
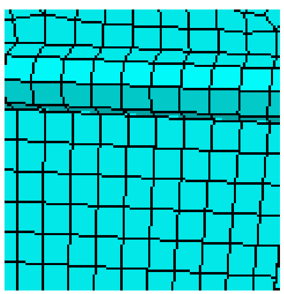
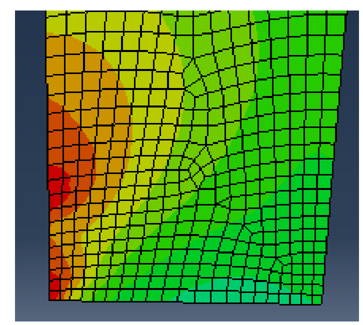
- Icons accelerate access to frequently used features



▶ Consistent environment

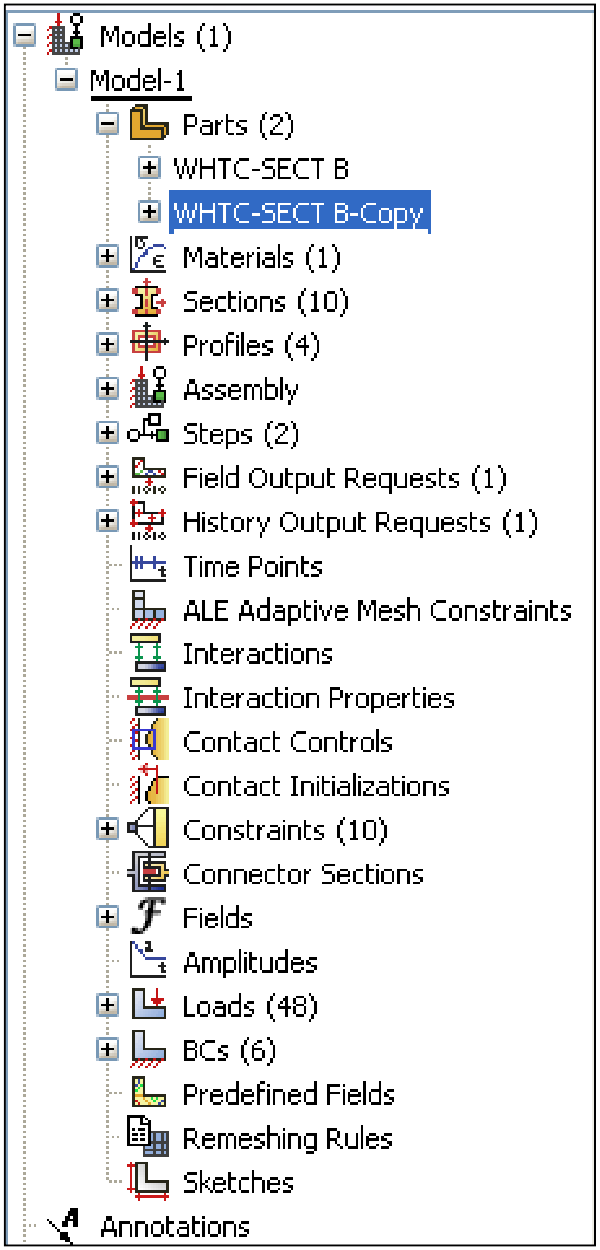
- Functionality is presented in modules
- Each module contains a logical subset of the overall functionality
- Once you understand the presentation of one module, you can easily understand the presentation of the other modules.



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

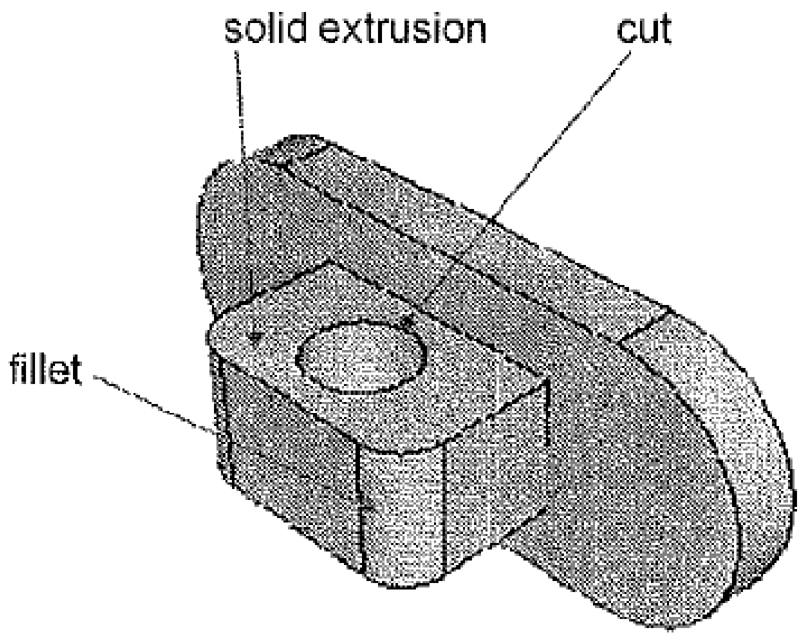
▶ Model Tree

- 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.



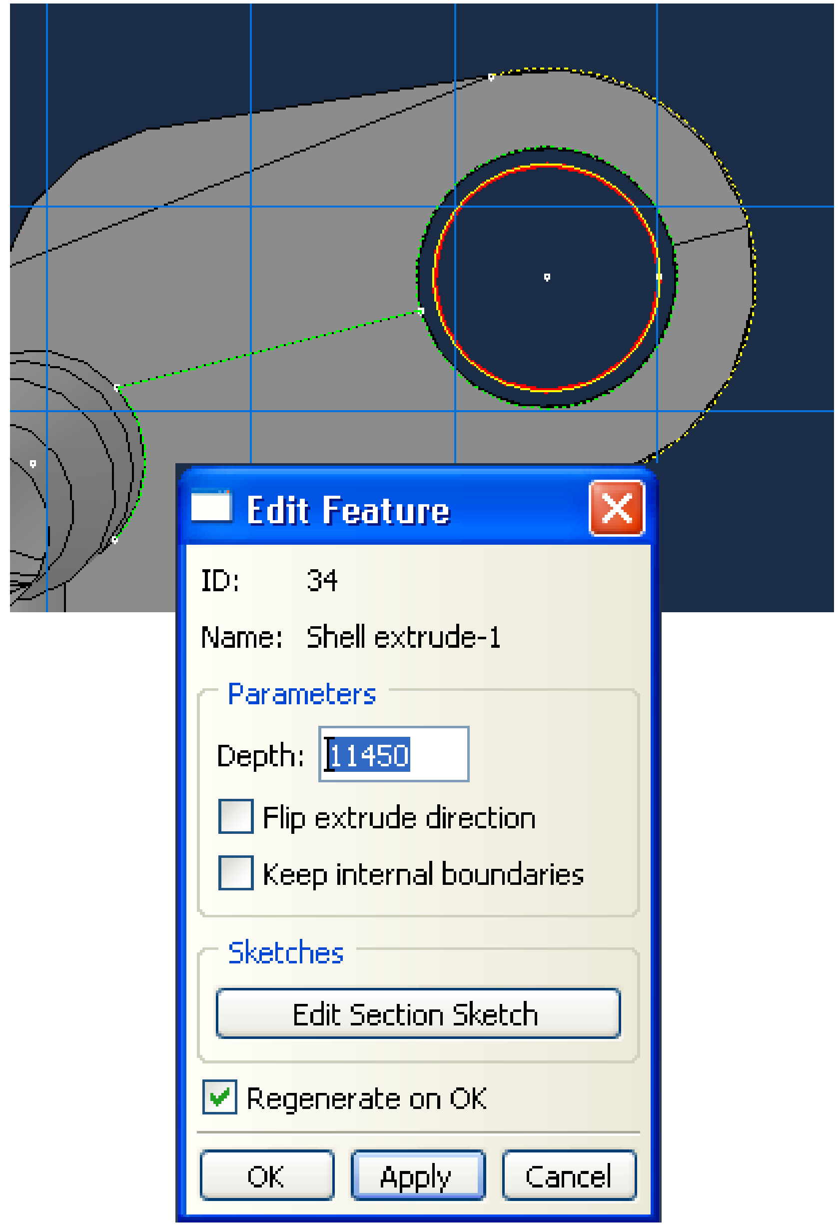
Part with several annotated features

- 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.

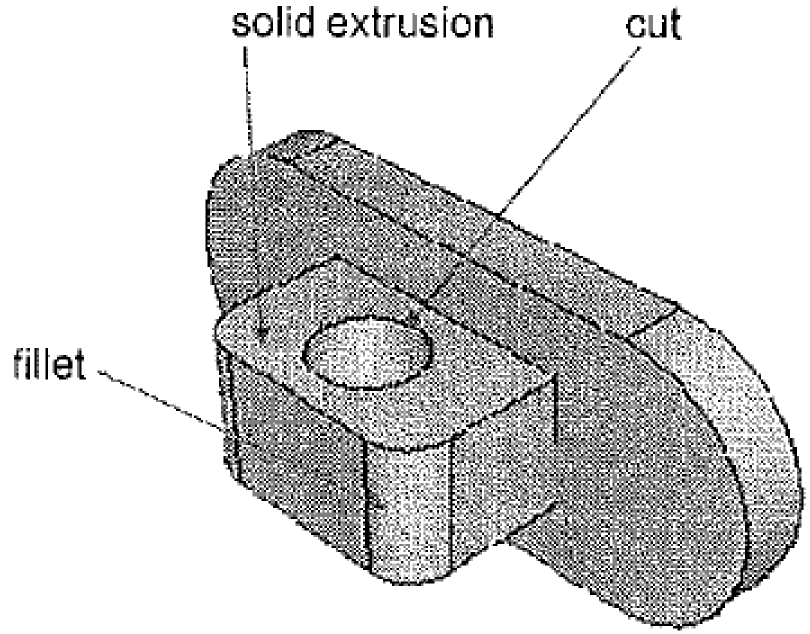
- Cut
  - Sketch of cut cross-section, depth of cut.

- 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

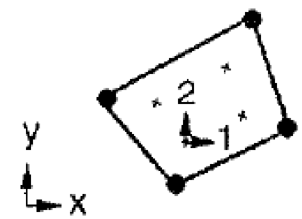
- ABAQUS uses no inherit sets of units
- It is the user's responsibility to use consistent units

▶ COORDINATE SYSTEMS

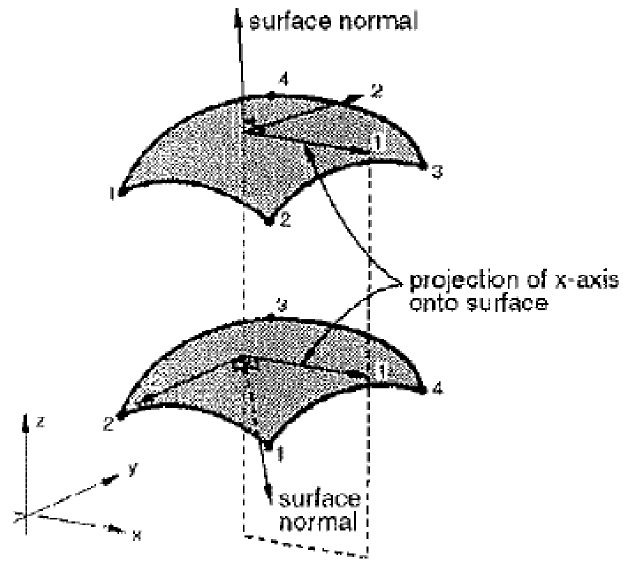
- For boundary conditions and point loads, the default coordinate system is the rectangular Cartesian system.
  - Alternative local rectangular, cylindrical, and spherical systems can be defined.
  - These local directions do not rotate with the material in large- displacement analyses.

▶ 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.



Default material directions for solid elements



Default material directions for shell and membrane elements

CHAPTER

II. Working with Geometry in ABAQUS

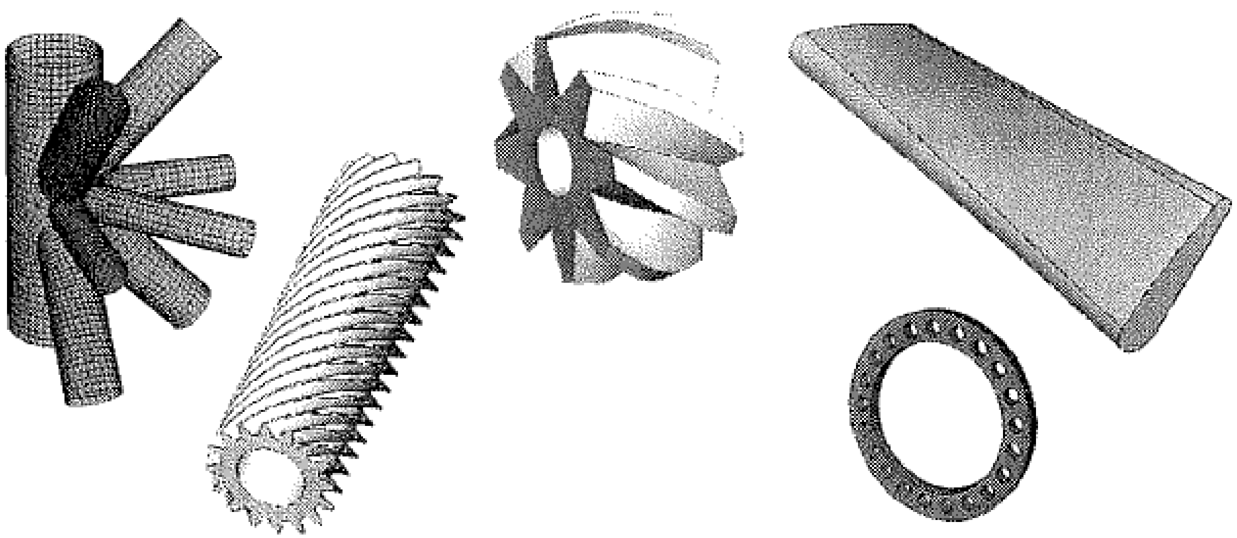
PARTS

► 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

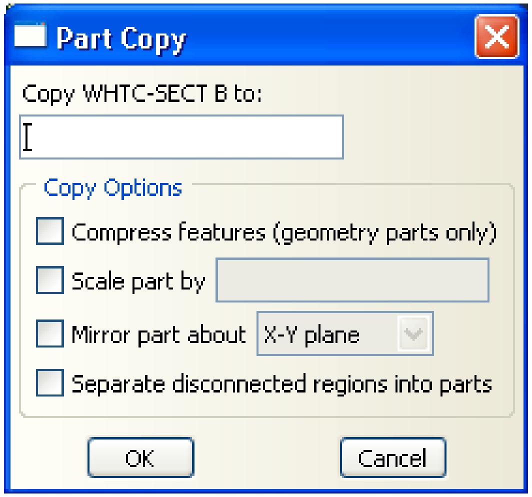
► Geometry imported from CAD systems

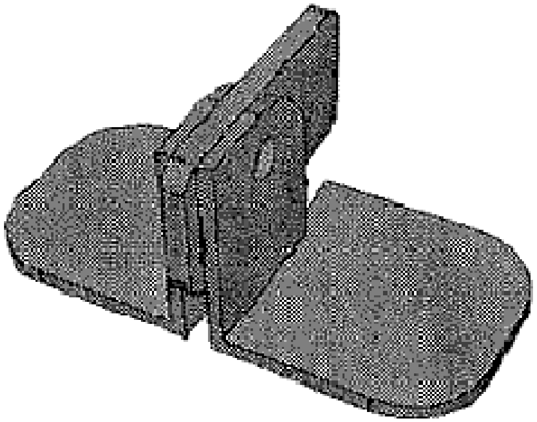
- Native CAD geometry
- Neutral formats
  - Imported geometry does not have a feature-based representation

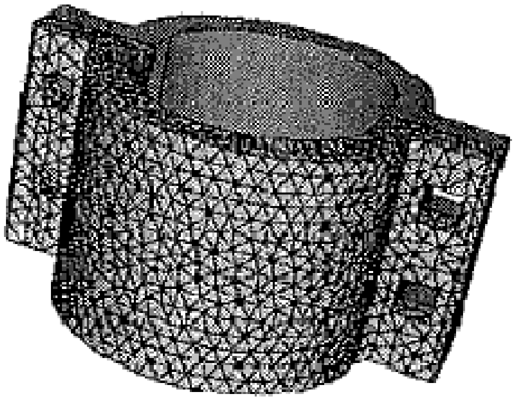


PARTS

- **Parts can be scaled and/or mirrored while copying**
  - This is available for both geometry and orphan meshes

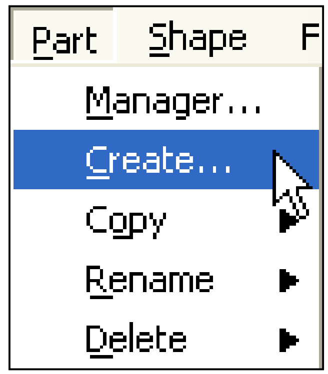


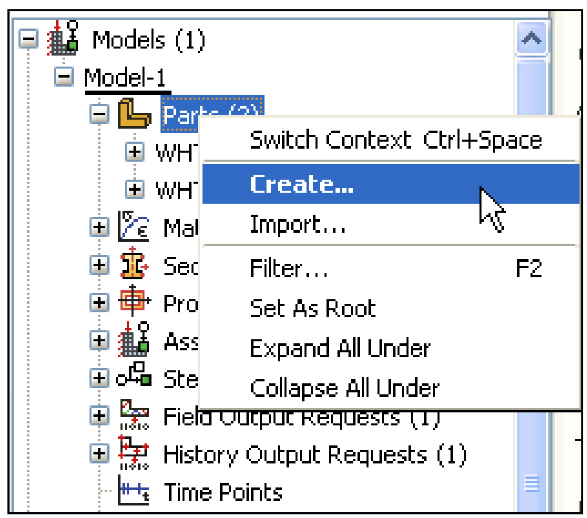


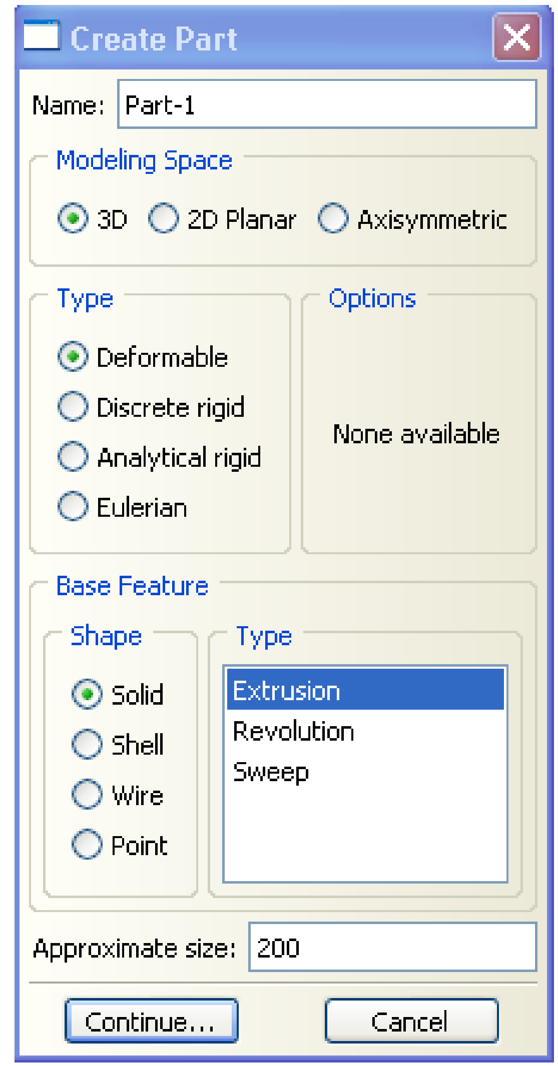


Building PART Using Part Module Tool

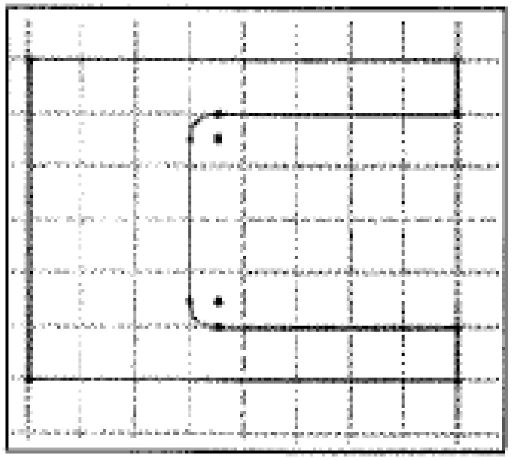
1. Select the appropriate dimensionality and part type—
   – Can be changed later.
2. Select the type of base feature—cannot be changed once selected.
   – The first feature you create while building a part is called the base feature.

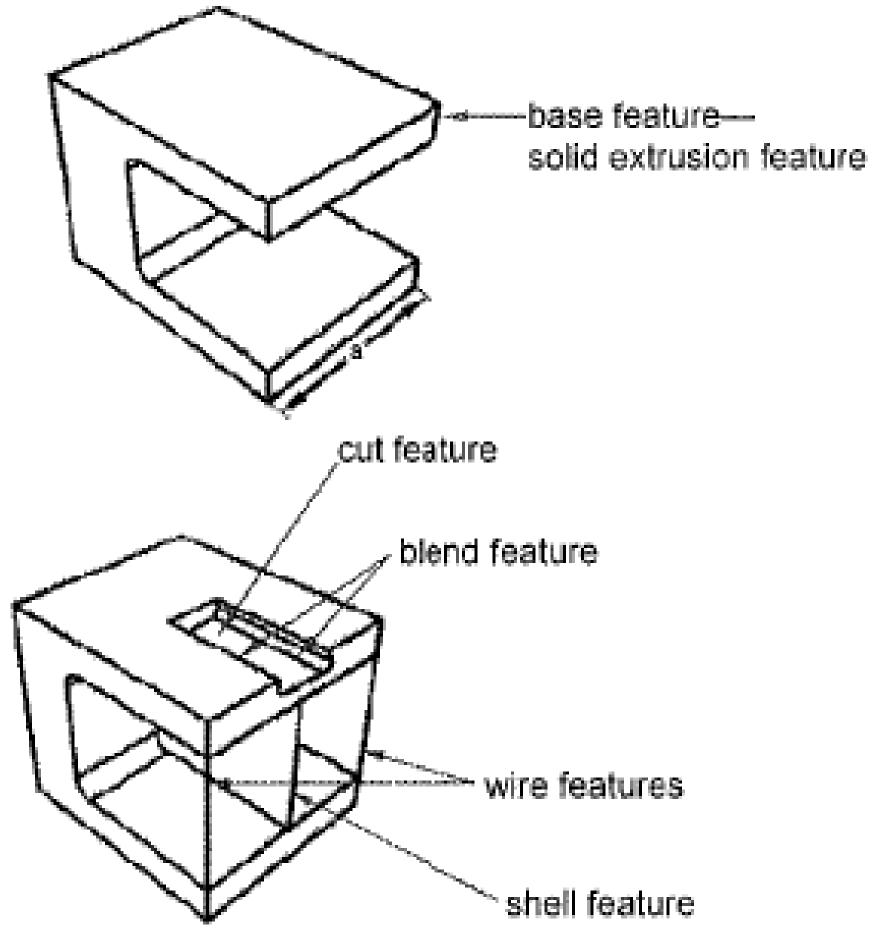






Building PART Using Part Module Tool



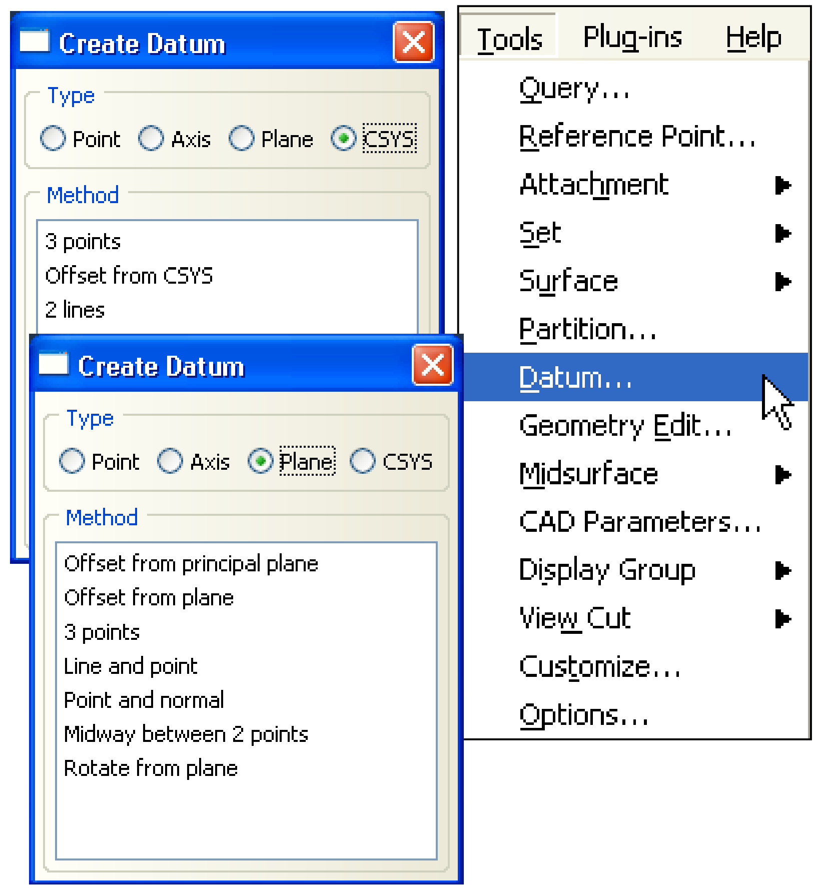


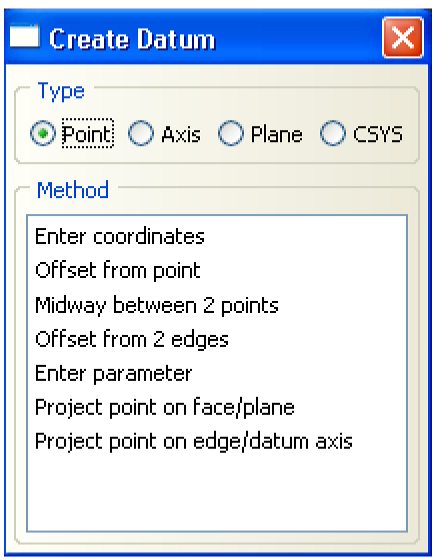
Base feature and
additional features

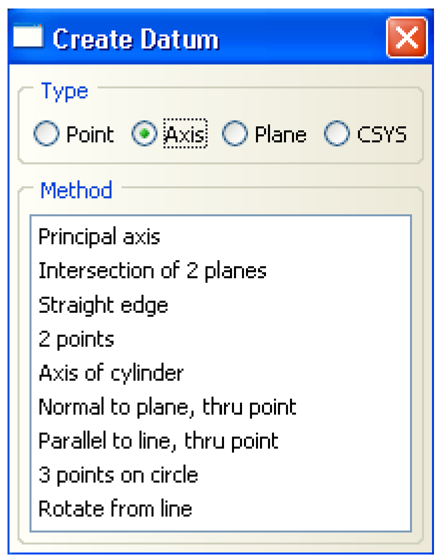
Building PART Using Part Module Tool

► ‘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







Building PART Using Part Module Tool

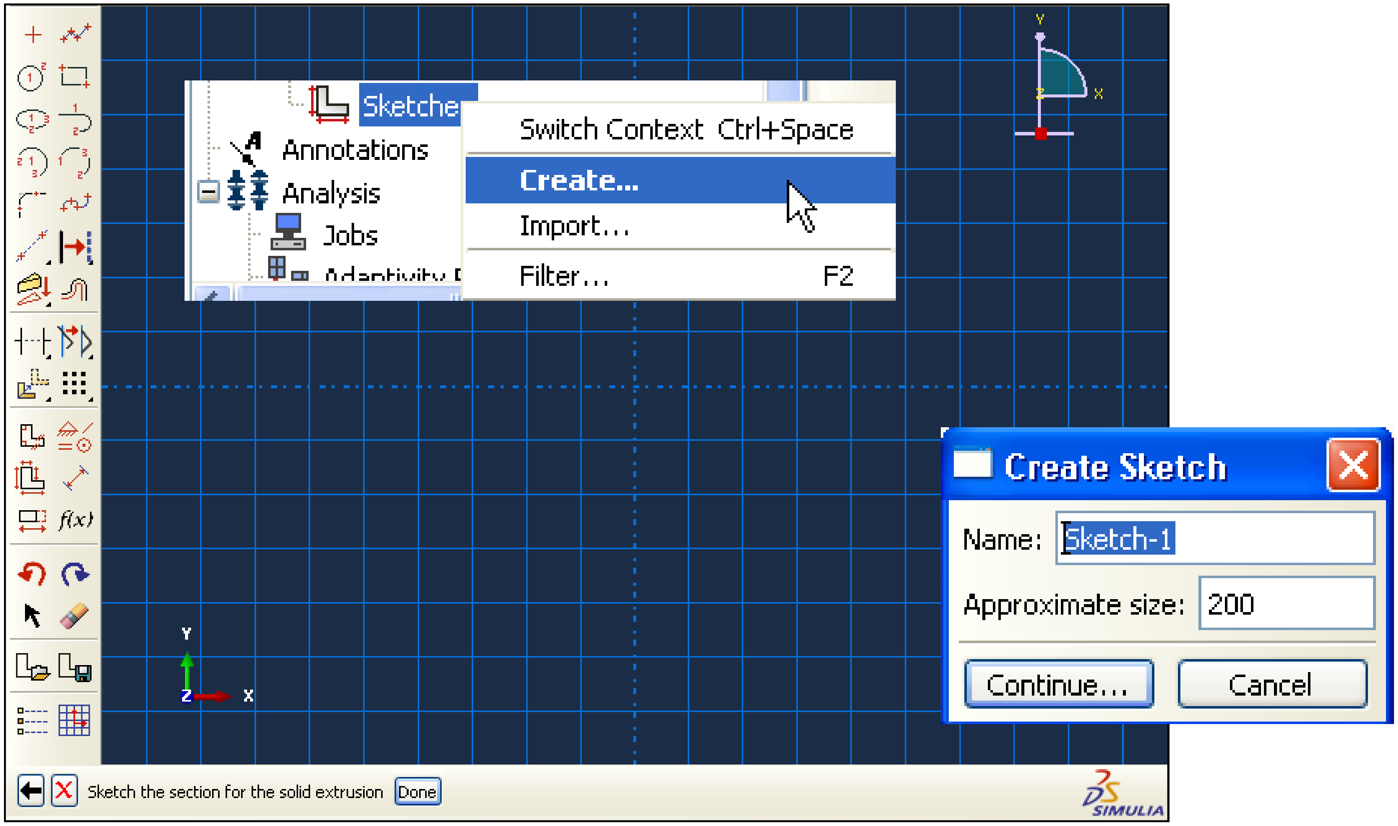
► Partitioning in the Part module

- 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

Building PART Using Part Module Tool

▶ Invoking the Sketcher

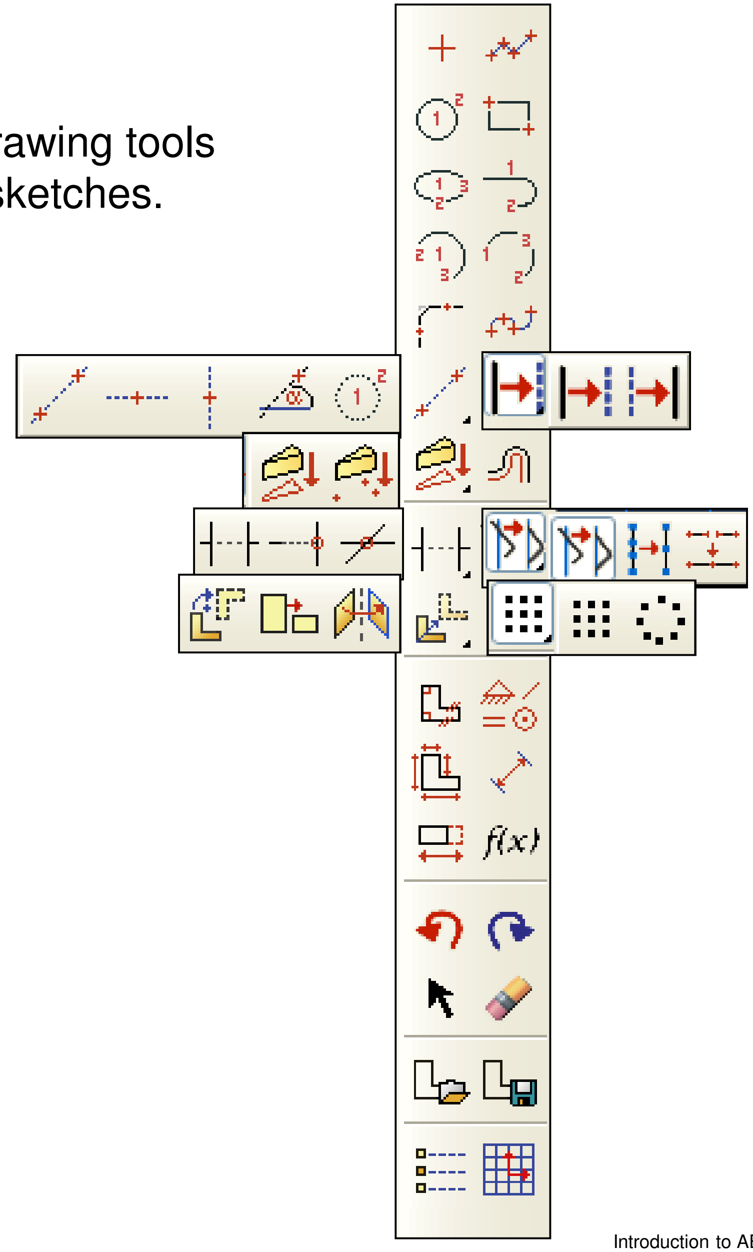
- When you create a base feature, ABAQUS/CAE automatically invokes the Sketcher with a blank sheet of virtual graph paper.
- An alternative is to create a “stand-alone” sketch using the Sketch module. Such sketches are not initially associated with a part but can be used later



Building PART Using Part Module Tool

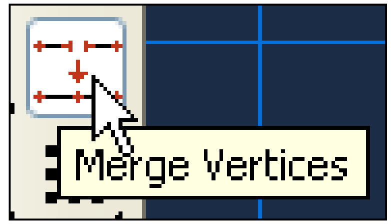
▶ Using the Sketcher

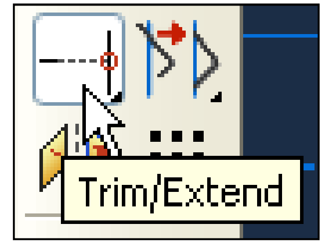
- The Sketcher toolset provides the basic drawing tools necessary to create moderately complex sketches.

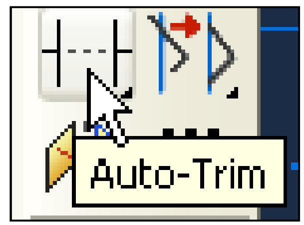


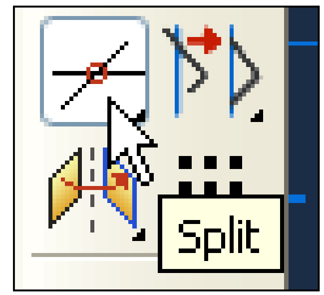
Building PART Using Part Module Tool

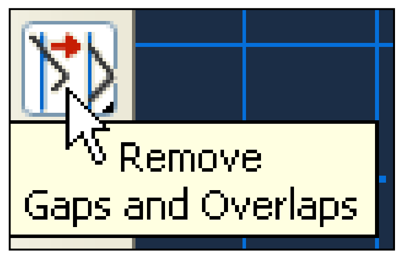
- **Trimming, extending, breaking, and merging edges**
  - Trim/extend edges
    - Breaks the edges at intersection
  - Auto-trim
    - Based on pre-selection
    - Does not break the intersecting edges
  - Break edges
    - Creates separate pieces with a common vertex
  - Merge edges
    - Close small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

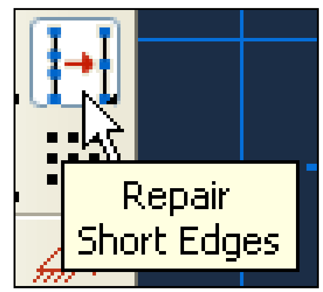








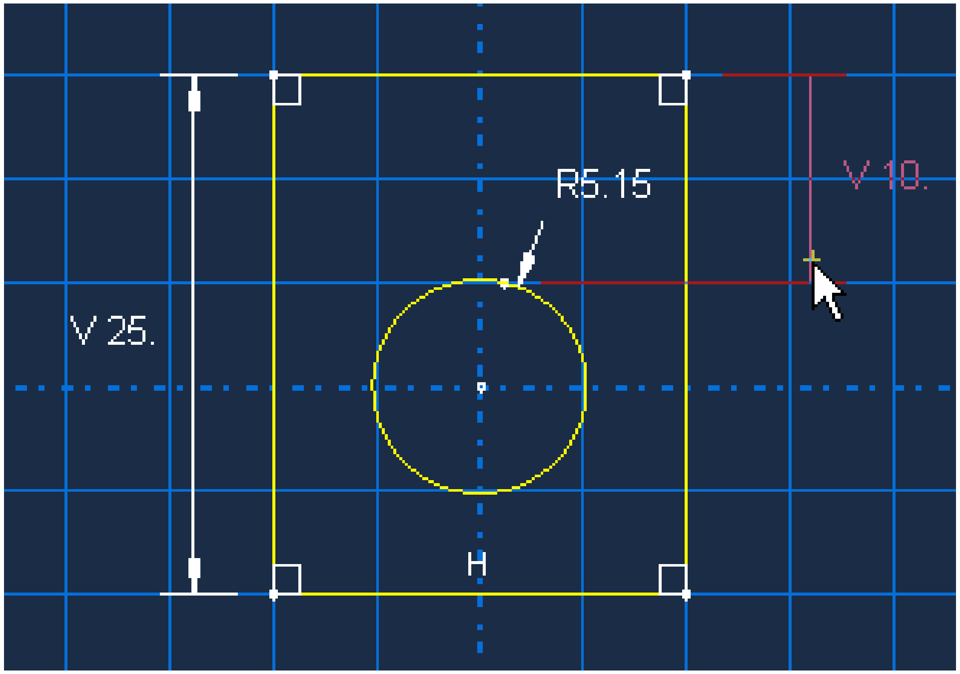




Building PART Using Part Module Tool

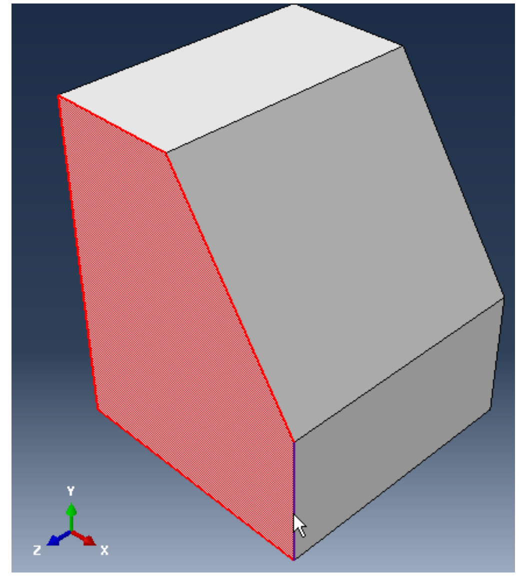
▶ Dimensions in the Sketcher

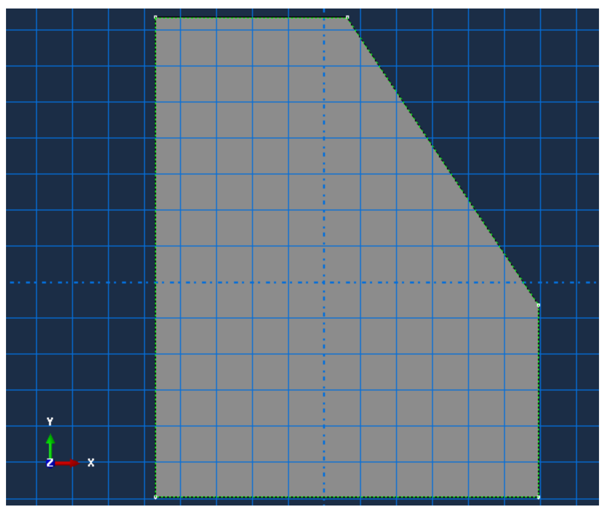
- After creating sketch geometry, you can add dimensions between geometric entities in the sketch
- You can refine the sketch by modifying the dimension values. The sketch actively changes to reflect the new dimension values
- Dimensions serve as comments for future reference.

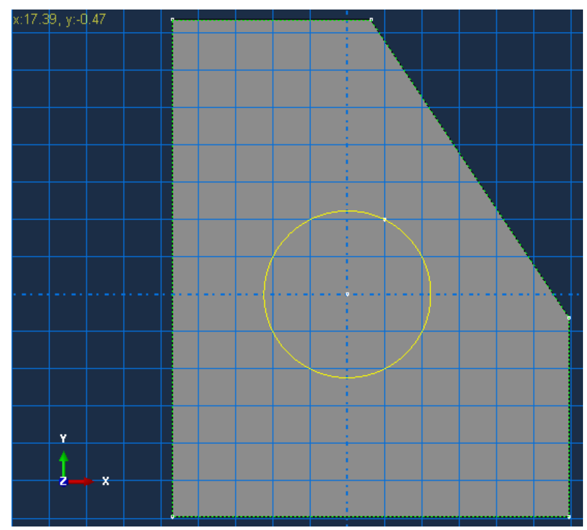


Building PART Using Part Module Tool

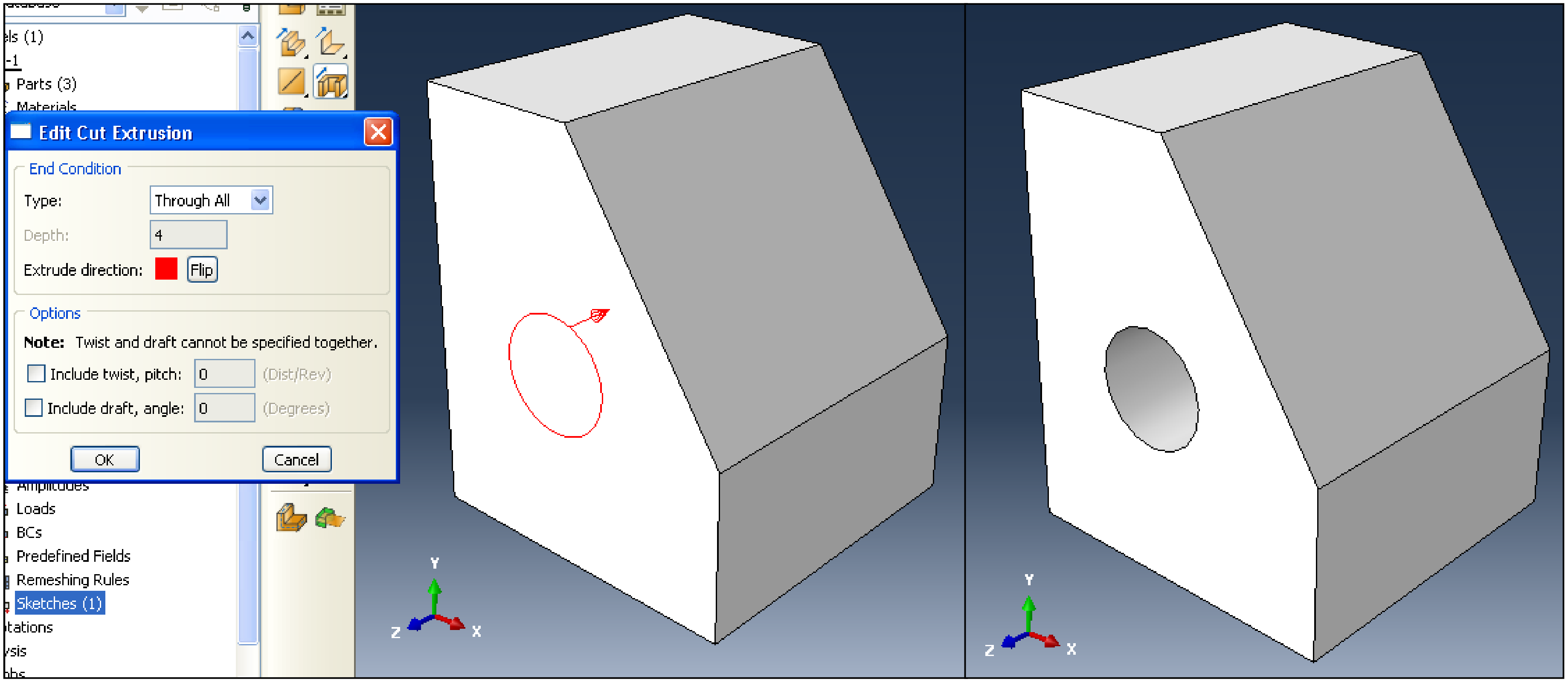
- **Adding features to a base feature**
  - When adding sketched features to a base feature, select the sketching plane from a planar face on the part or a datum plane.
  - You can control the sketch orientation by selecting a planar face and an edge.
  - By default, the edge will appear vertical and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





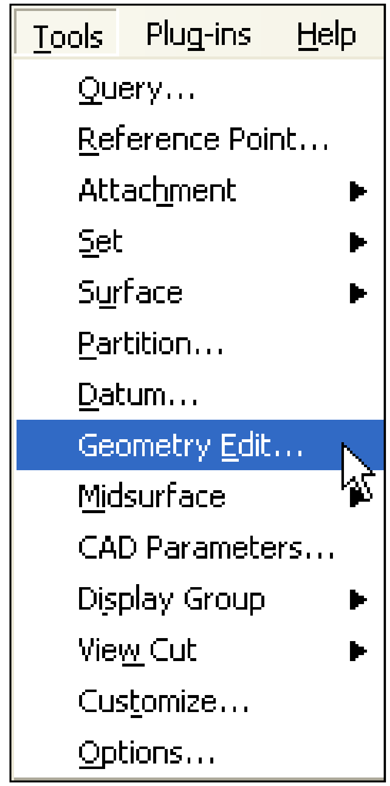


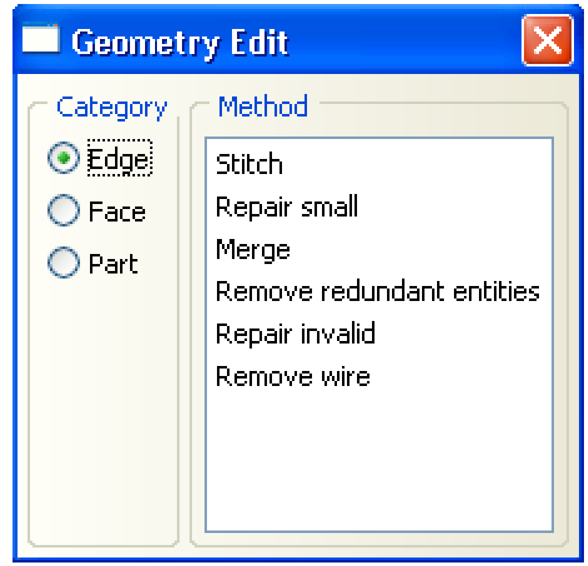
Building PART Using Part Module Tool

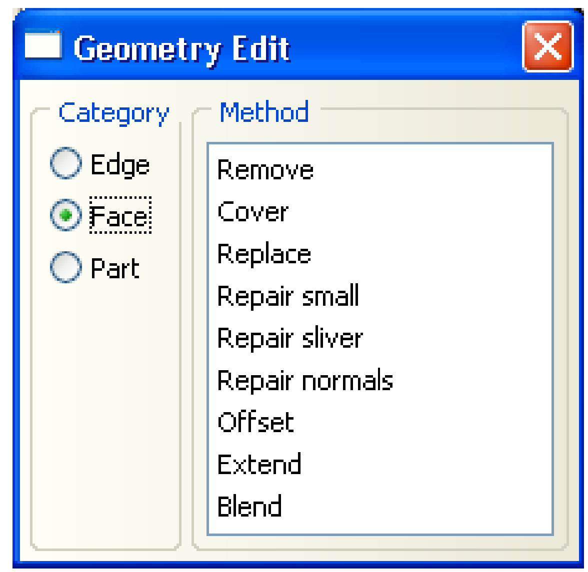


Geometry Repair

- **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.





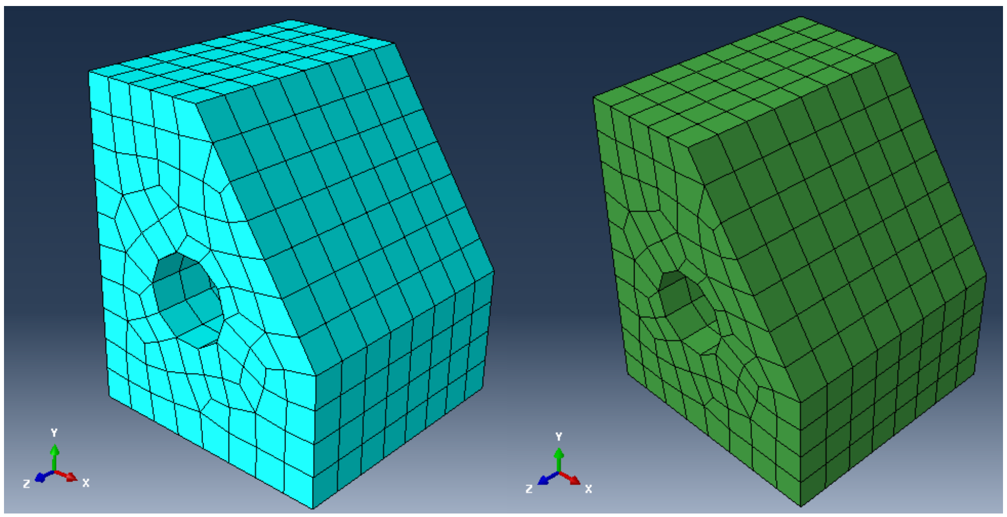


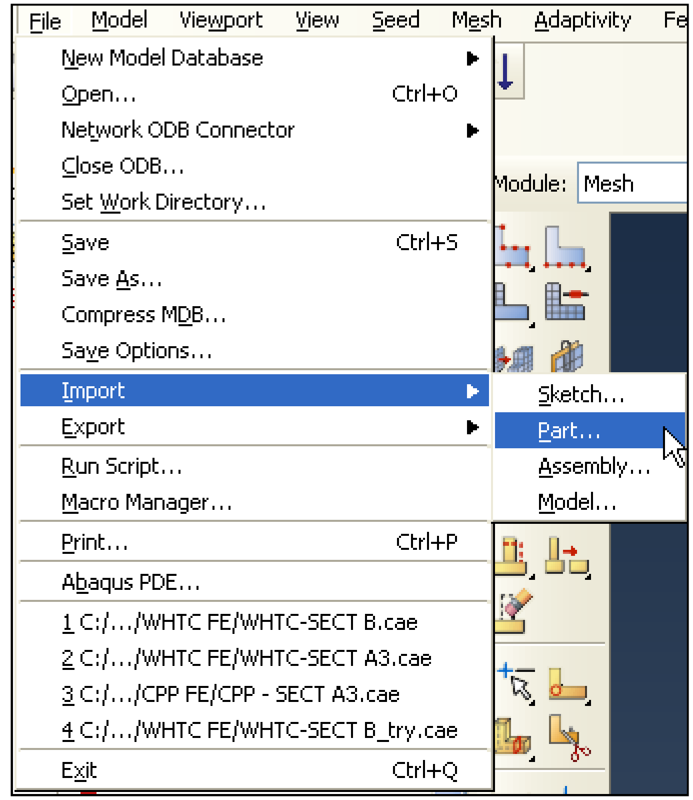
CONTENTS

III. Working with models Created Outside ABAQUS

Orphan Mesh Import

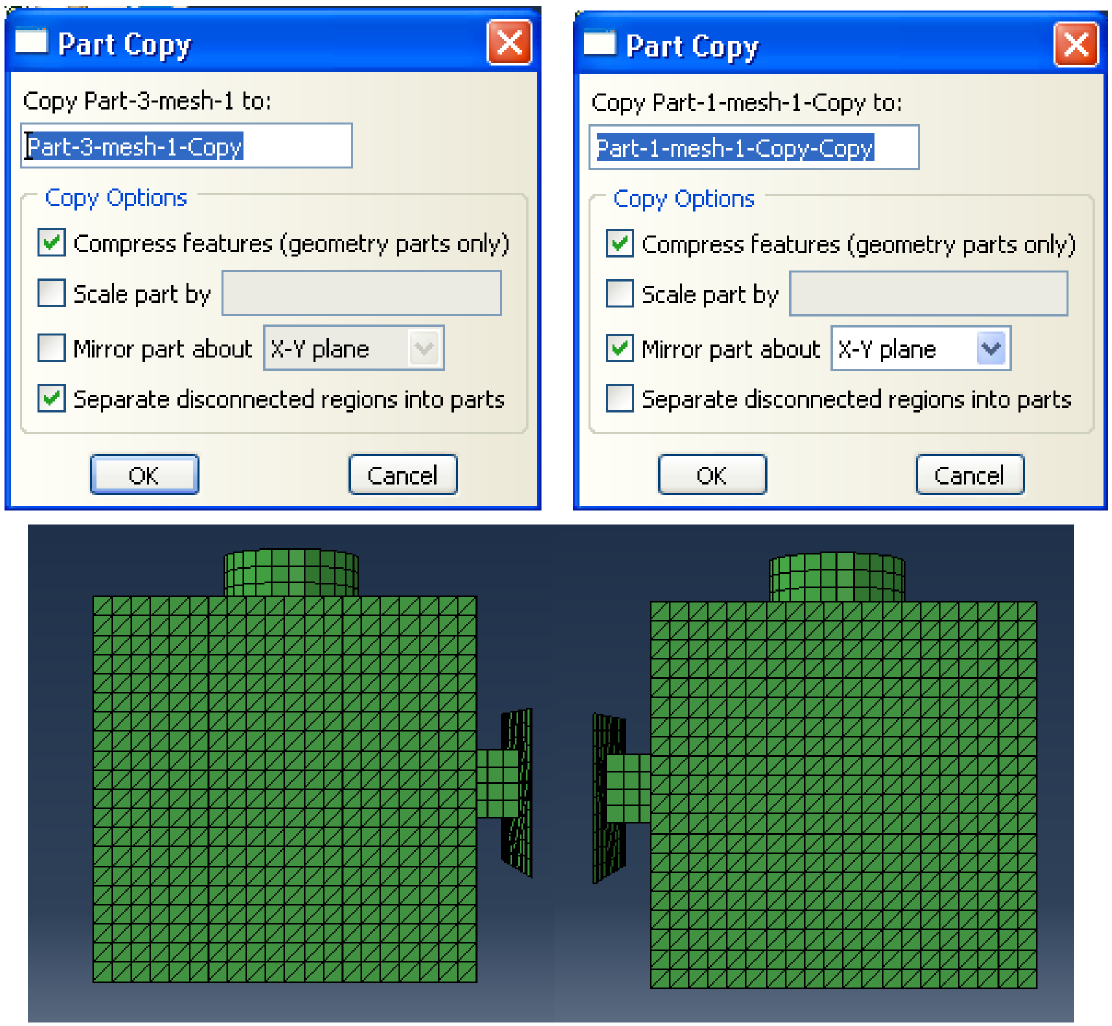
- An existing mesh can be imported from an ABAQUS input (.inp) or output database (.odb) file
- The mesh is called an orphan mesh because it has no associated parent geometry



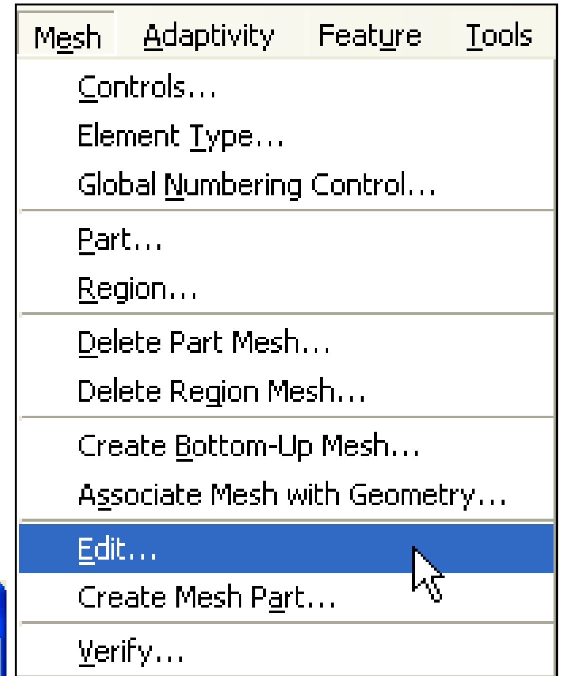


Orphan Mesh Import

- **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.

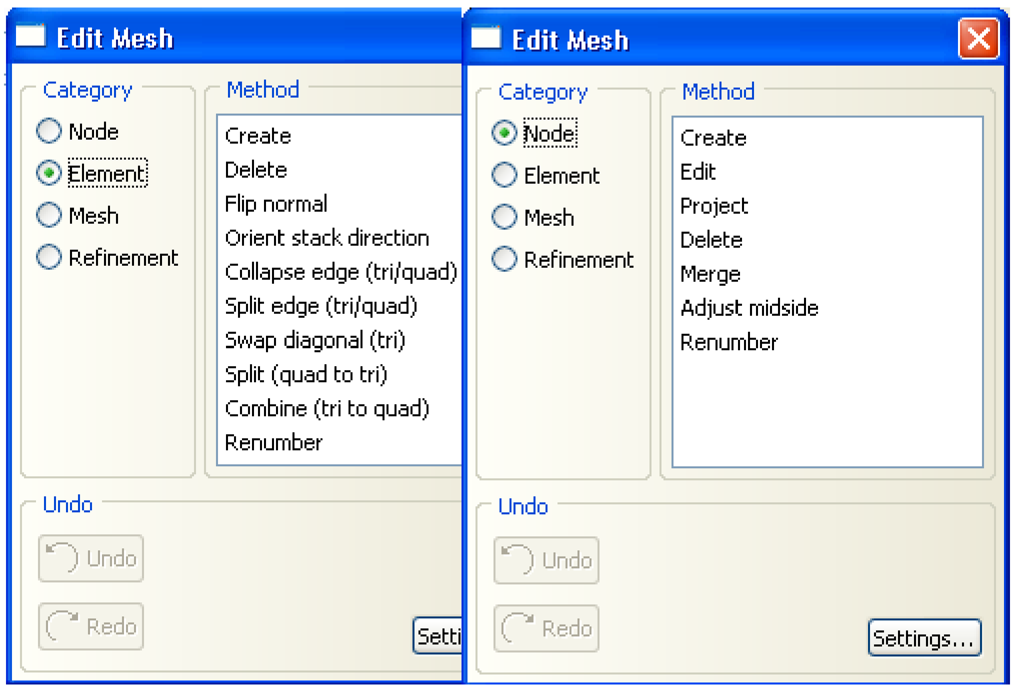


Orphan Mesh Import



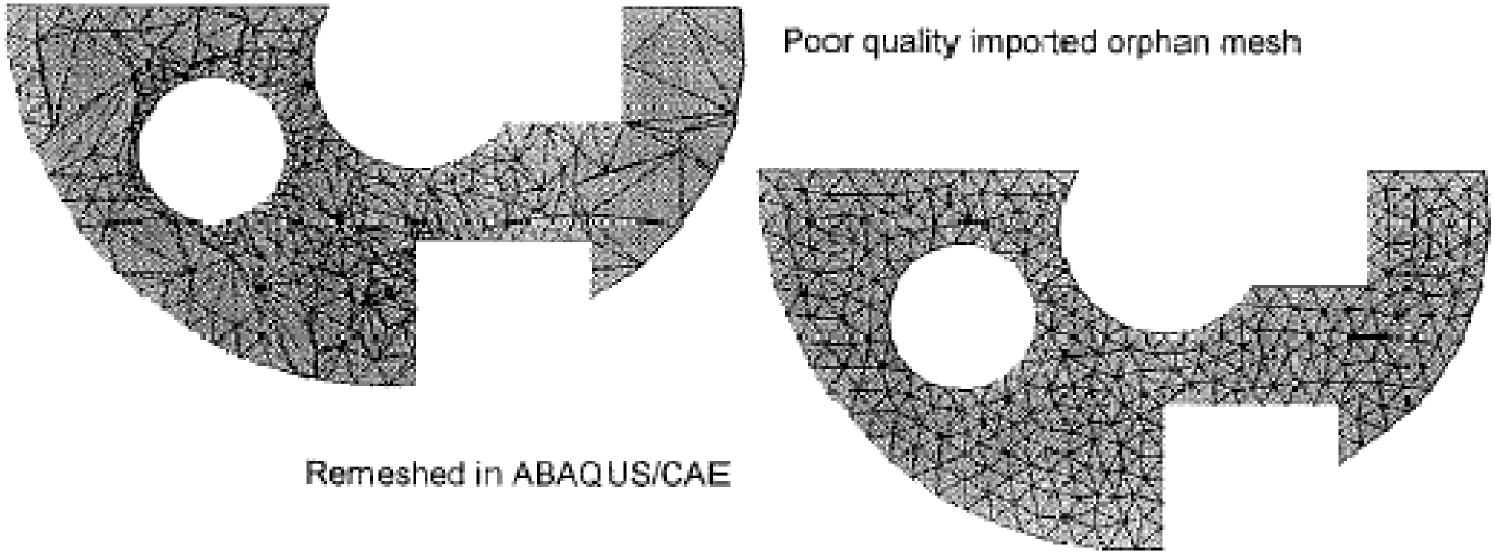
▶ Orphan mesh editing

- 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



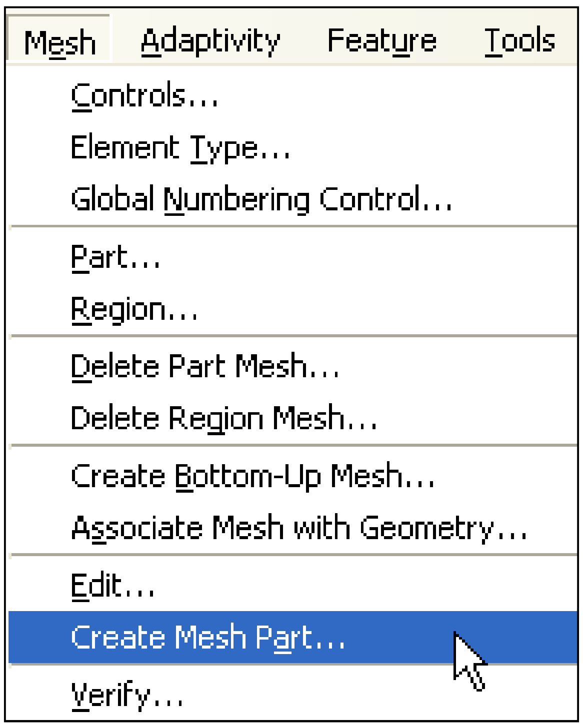
Orphan Mesh Import

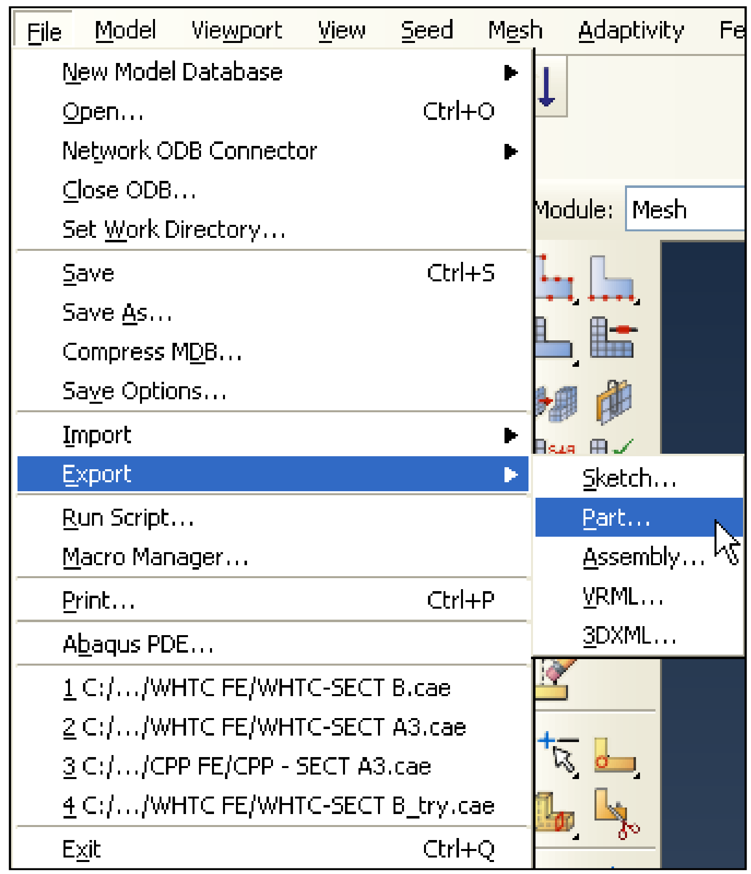
- **Re-meshing of planar orphan meshes**
  - Mesh consists of either first or second order triangular elements; re-meshed based on boundary sizes or new uniform global size.
  - Overall material area is preserved.



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.





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

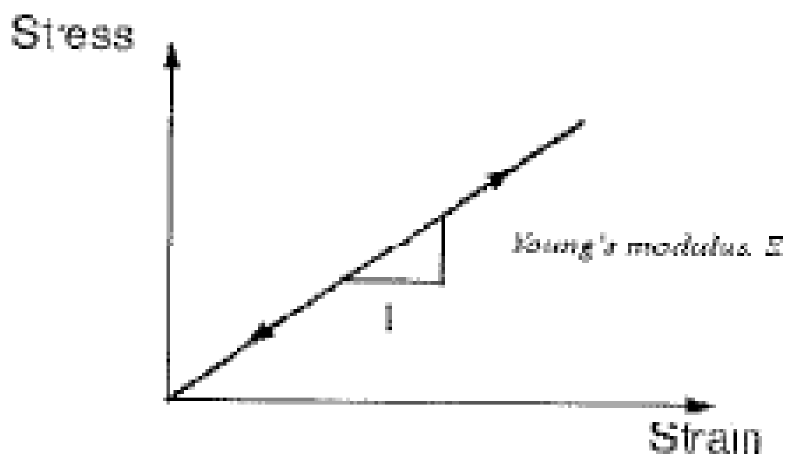
- **For a linear elastic material, Hooke's law states**

  stress α strain

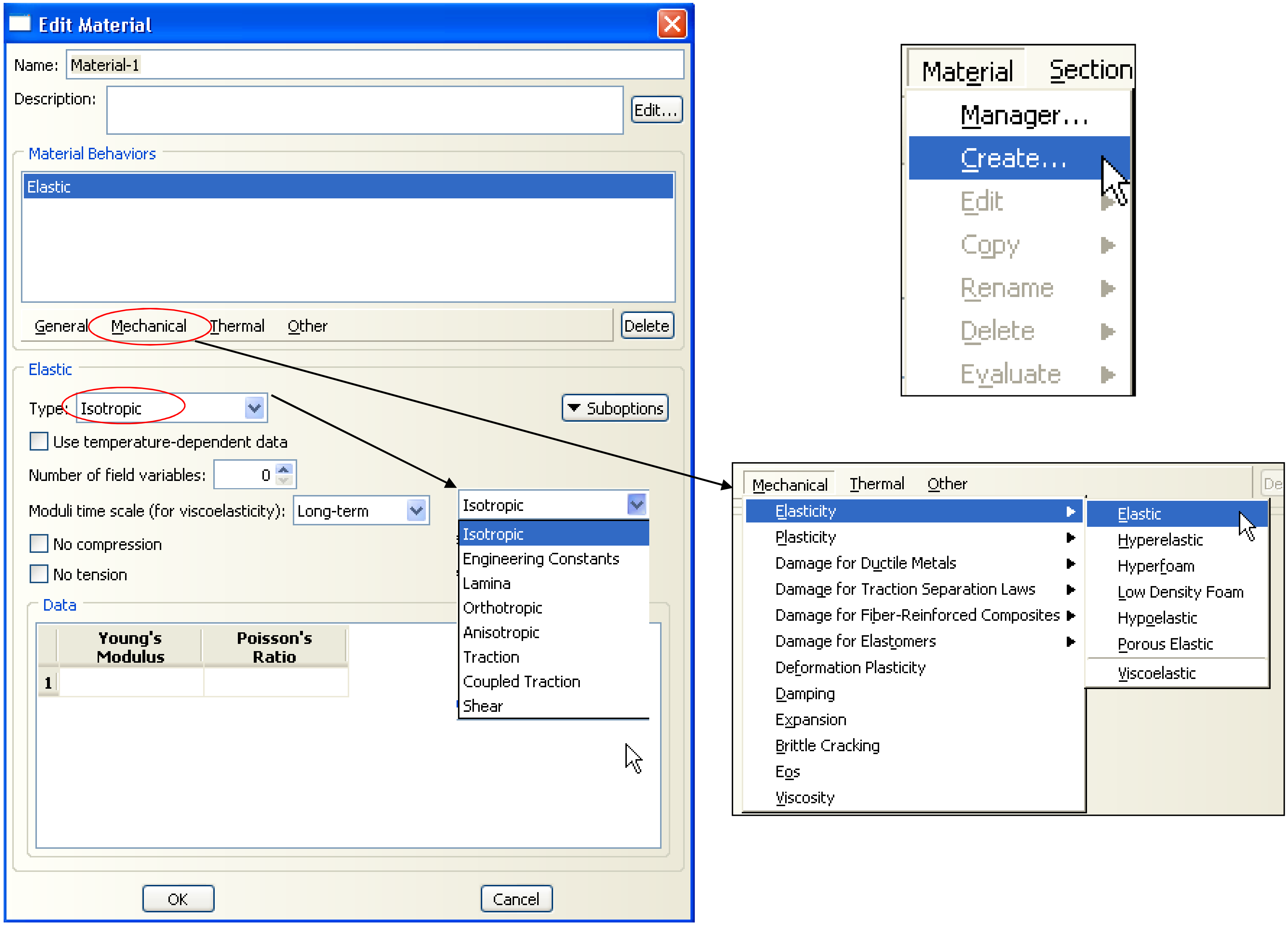
- **The generalized form of the law is written as**

  $$\sigma = D^{el} : \varepsilon^{el}$$

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

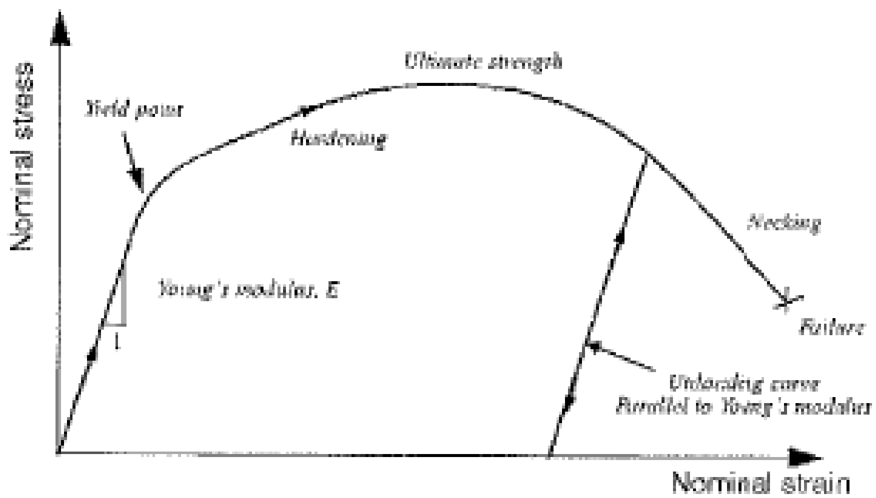


Uniaxial stress-strain curve for a linear elastic material


Edit Material
Name: Material-1
Description:
Edit...
Material Behaviors
Elastic
General Mechanical Thermal Other
Delete
Elastic
Type: Isotropic
Suboptions
Use temperature-dependent data
Number of field variables: 0
Moduli time scale (for viscoelasticity): Long-term
No compression
No tension
Data
Young's Modulus
Poisson's Ratio
1
OK
Cancel
Isotropic
Isotropic
Engineering Constants
Lamina
Orthotropic
Anisotropic
Traction
Coupled Traction
Shear
Material Section
Manager...
Create...
Edit
Copy
Rename
Delete
Evaluate
Mechanical Thermal Other
Elasticity
Plasticity
Damage for Ductile Metals
Damage for Traction Separation Laws
Damage for Fiber-Reinforced Composites
Damage for Elastomers
Deformation Plasticity
Damping
Expansion
Brittle Cracking
Eos
Viscosity
Elastic
Hyperelastic
Hyperfoam
Low Density Foam
Hypoelastic
Porous Elastic
Viscoelastic

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**

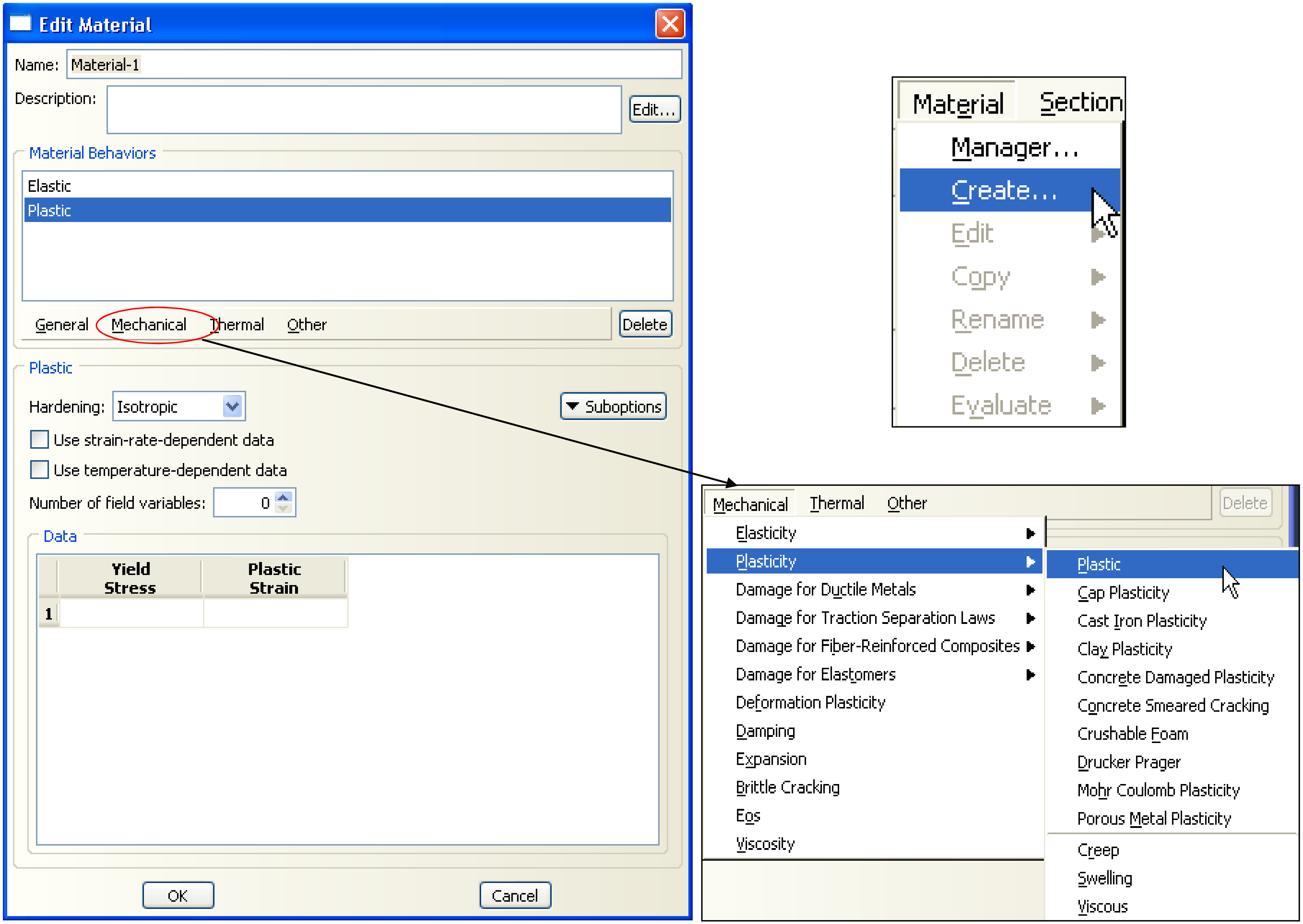
- **If the material is strained beyond yield and the strain is then reversed, the material immediately recovers its elastic stiffness.**

Metal Plasticity

► Hardening

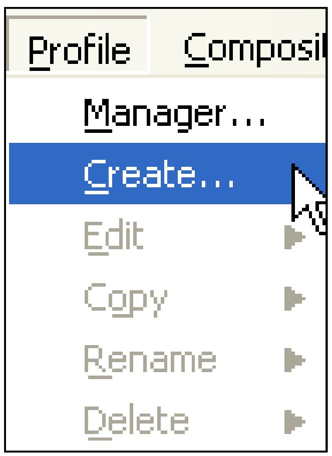
- The yield surface may change as a result of plastic deformation. The change in the yield surface is defined by the hardening law.
- The following hardening laws are available in ABAQUS:

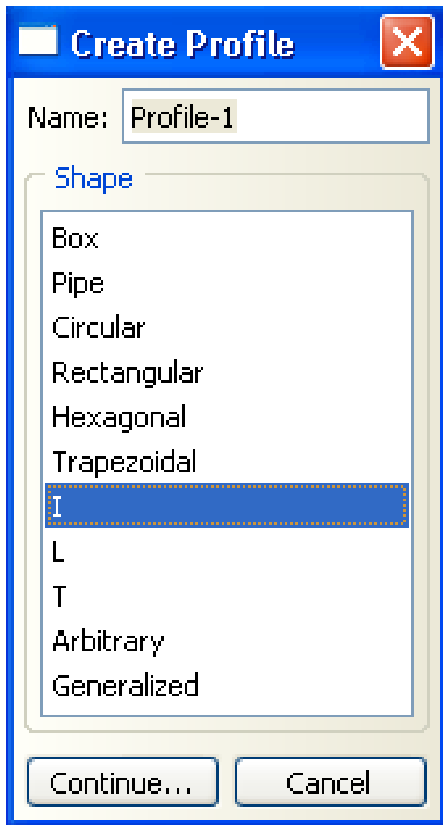
– Perfect plasticity – Isotropic hardening	*Intended for applications such as crash analyses, metal forming, and general collapse studies*
– Kinematic hardening – Combined isotropic/kinematic hardening	*Intended for applications involving cyclic loading*
– Johnson-Cook plasticity	*Well suited to model high-strain rate deformation of metals; only available in ABAQUS/Explicit*

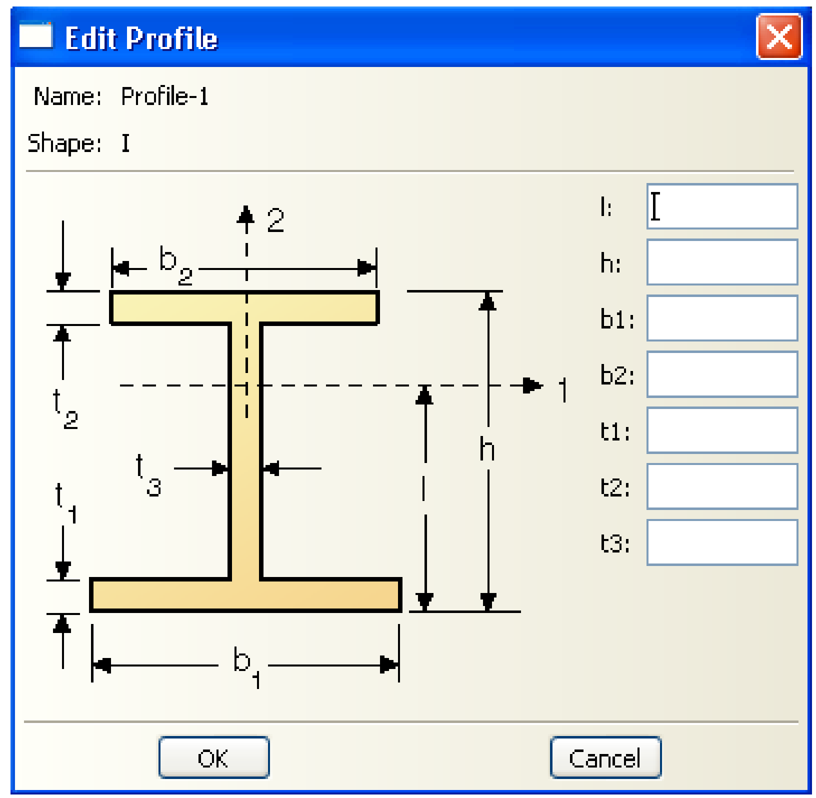

Edit Material
Name: Material-1
Description:
Edit...
Material Behaviors
Elastic
Plastic
General
Mechanical
Thermal
Other
Delete
Plastic
Hardening: Isotropic
Suboptions
Use strain-rate-dependent data
Use temperature-dependent data
Number of field variables: 0
Data
Yield Stress
Plastic Strain
1
OK
Cancel
Material
Section
Manager...
Create...
Edit
Copy
Rename
Delete
Evaluate
Mechanical
Thermal
Other
Delete
Elasticity
Plasticity
Damage for Ductile Metals
Damage for Traction Separation Laws
Damage for Fiber-Reinforced Composites
Damage for Elastomers
Deformation Plasticity
Damping
Expansion
Brittle Cracking
Eos
Viscosity
Plastic
Cap Plasticity
Cast Iron Plasticity
Clay Plasticity
Concrete Damaged Plasticity
Concrete Smeared Cracking
Crushable Foam
Drucker Prager
Mohr Coulomb Plasticity
Porous Metal Plasticity
Creep
Swelling
Viscous

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.

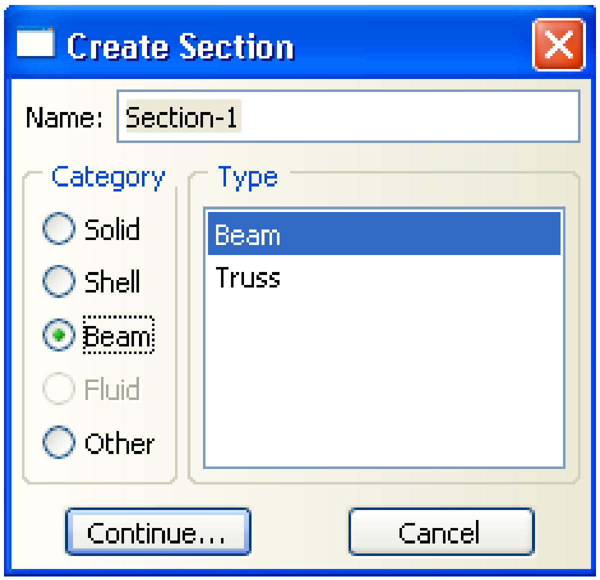


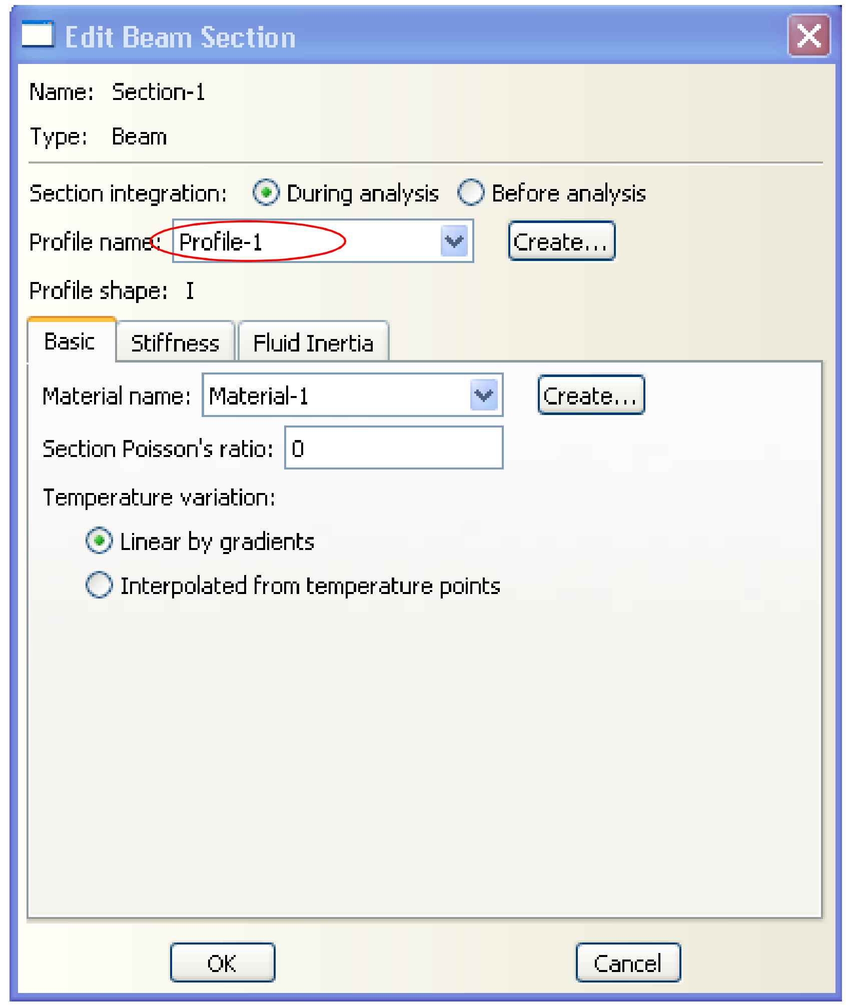




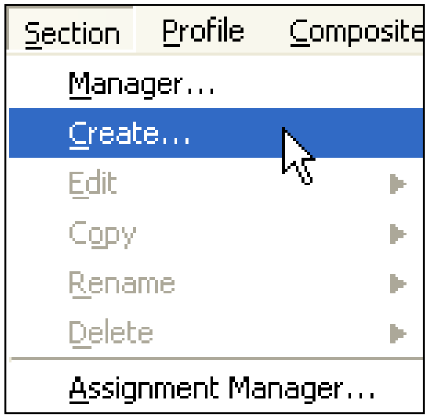
Section Properties

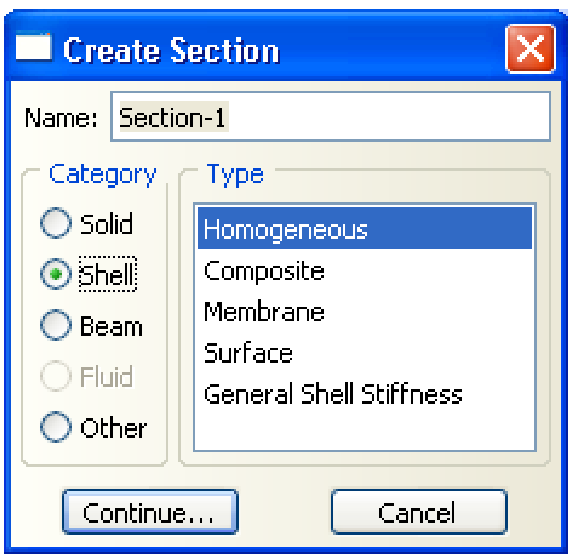
- **For beams and shells the stiffness of the section is computed numerically by integrating the section properties**
  - The stiffness may be computed either before the analysis (done once, efficient for linear analysis) or during the analysis

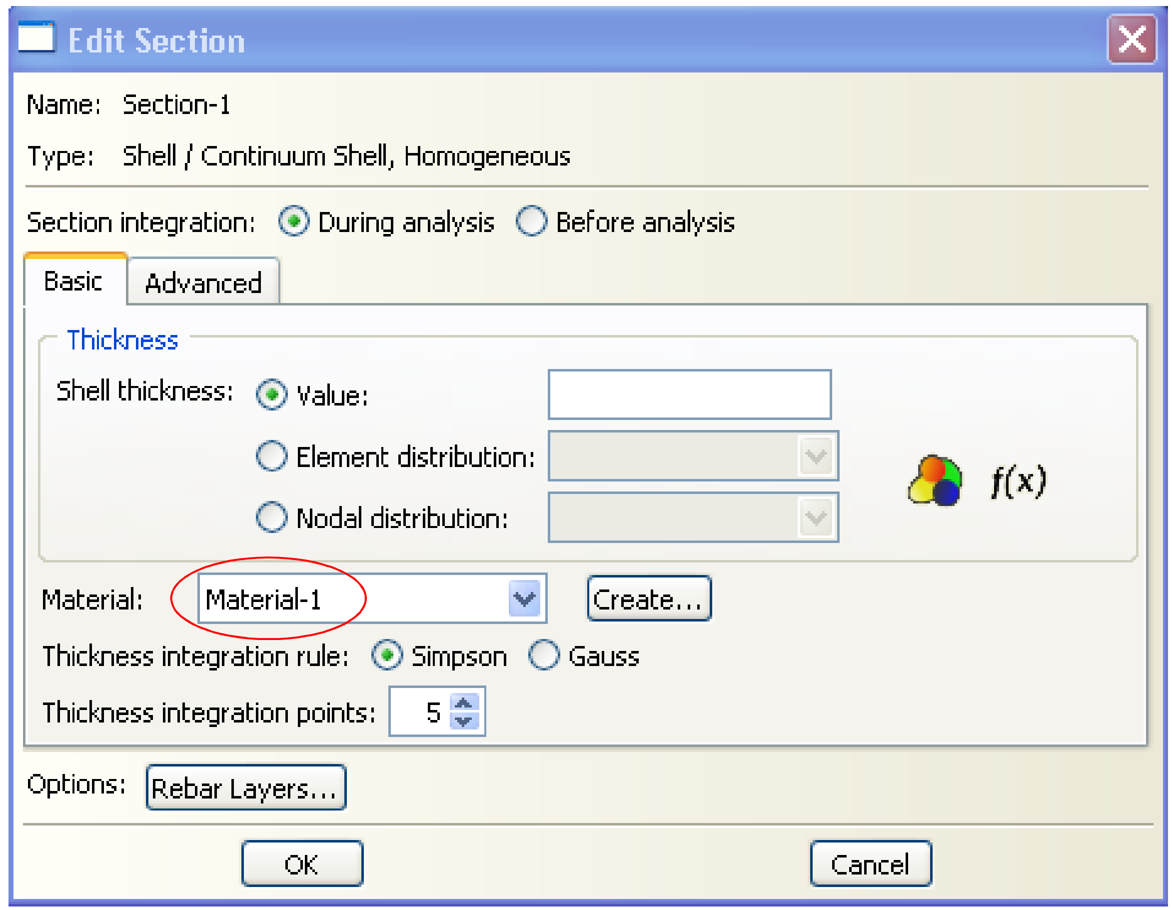




Section Properties







CHAPTER

II. Assemblies in ABAQUS

What is an Assembly?

- 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.

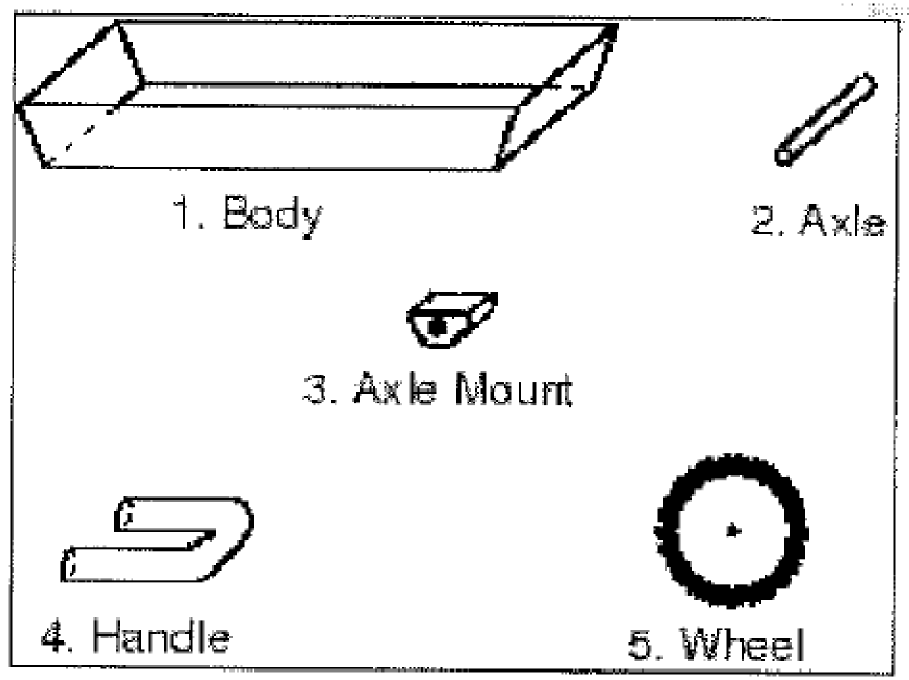
- For convenience, instances can be turned on and off for viewing.

- The following points explain the relationship between parts, part instances, and assemblies.

What is an Assembly?

▶ 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

What is an Assembly?

▶ Part instances and assemblies

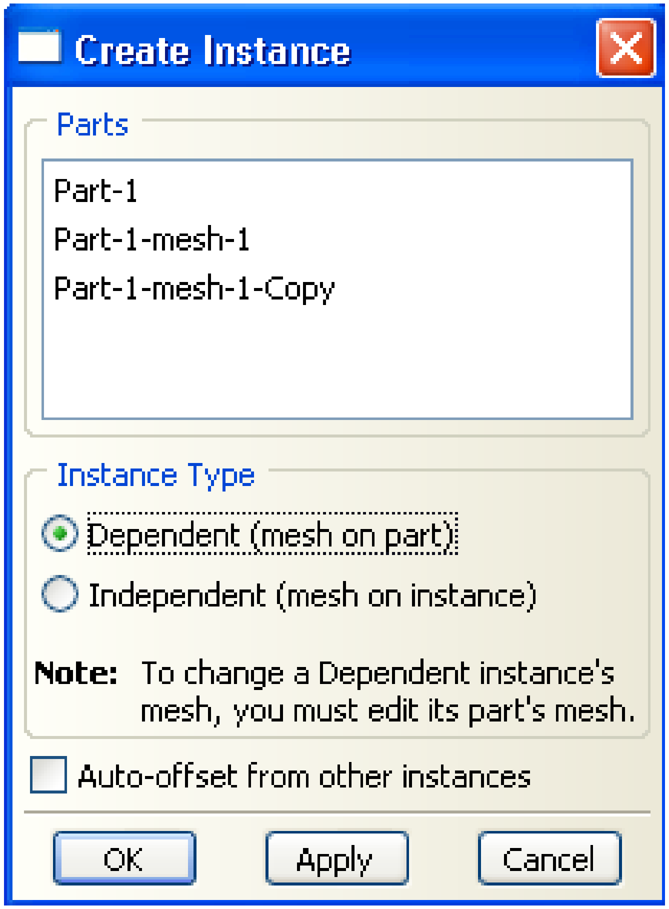
- You create instances of your parts in the Assembly module
  - An instance always maintains its association with the original part
  - You can create instance of a part many times and assemble multiple instances of the same part
  - Each instance of the part is associated with the section properties 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.

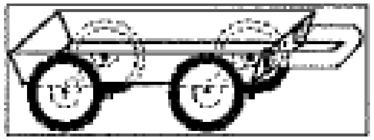
What is an Assembly?

- **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

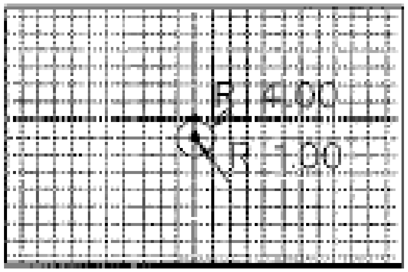


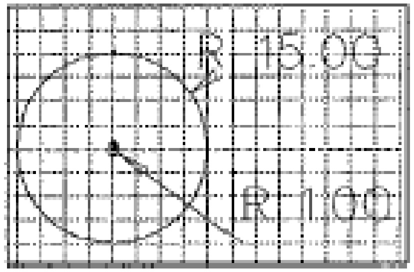
What is an Assembly?

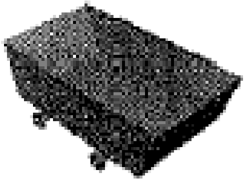
- **Example: Assembly of a child's wagon.**



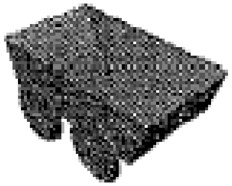
One instance of the body
Two instances of the axle
Four instances of the axle mount
One instance of the handle
Four instances of the wheel





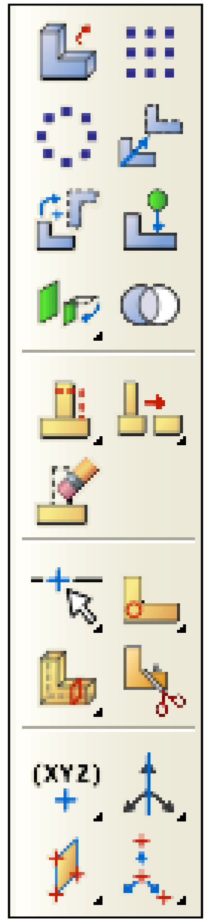


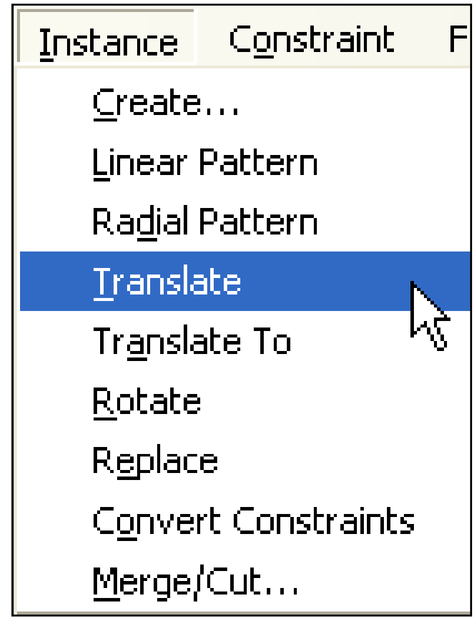
The wheel instances are automatically updated when the part definition changes.



Positioning Part instances

- **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



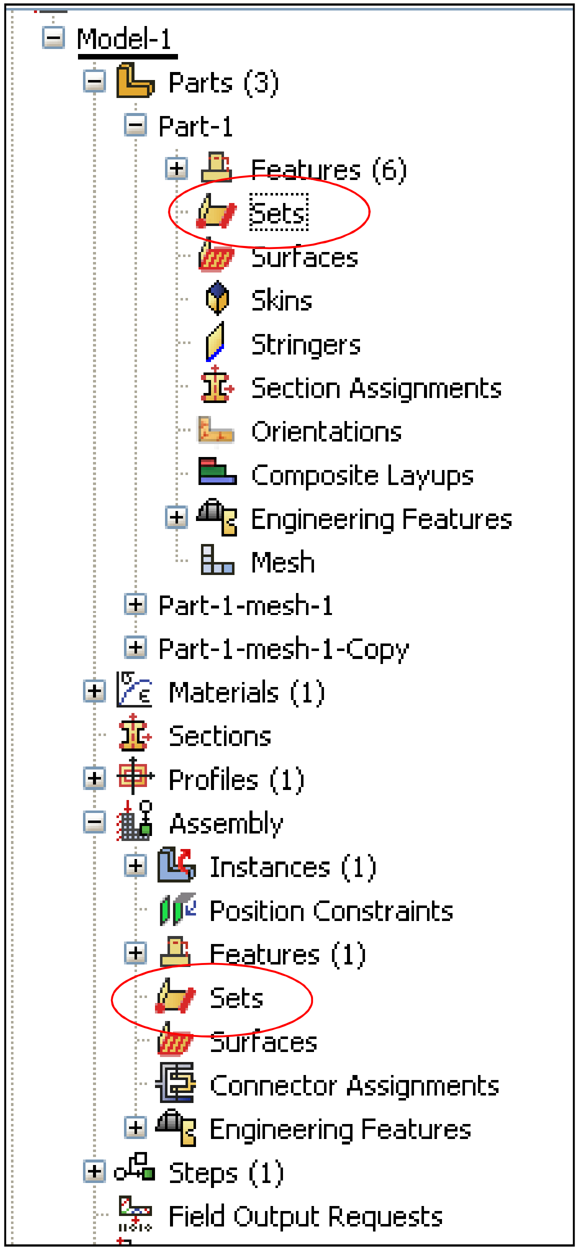


Sets and Surfaces

- **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

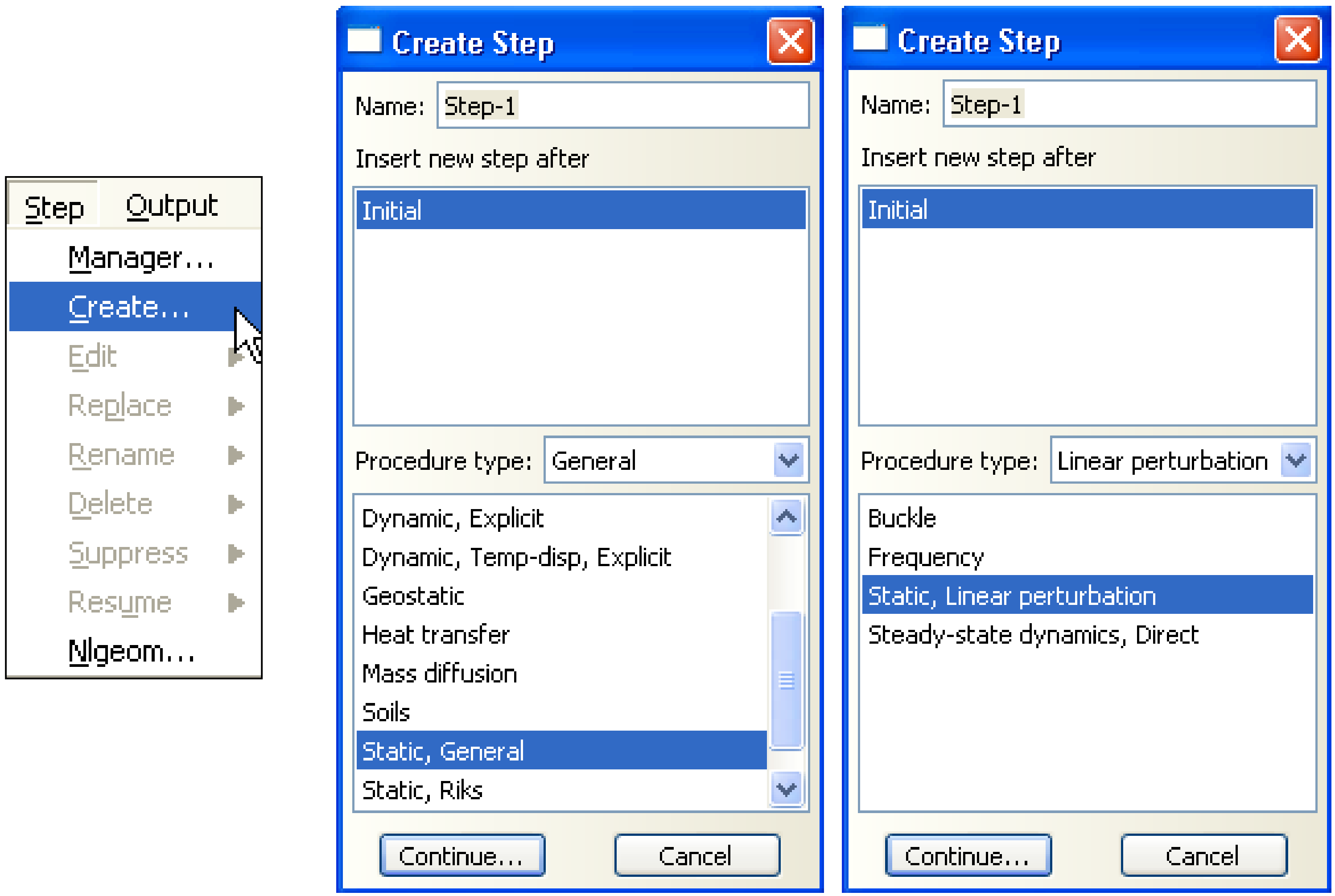
- **ABAQUS distinguishes between two kinds of analysis procedure:**

- **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

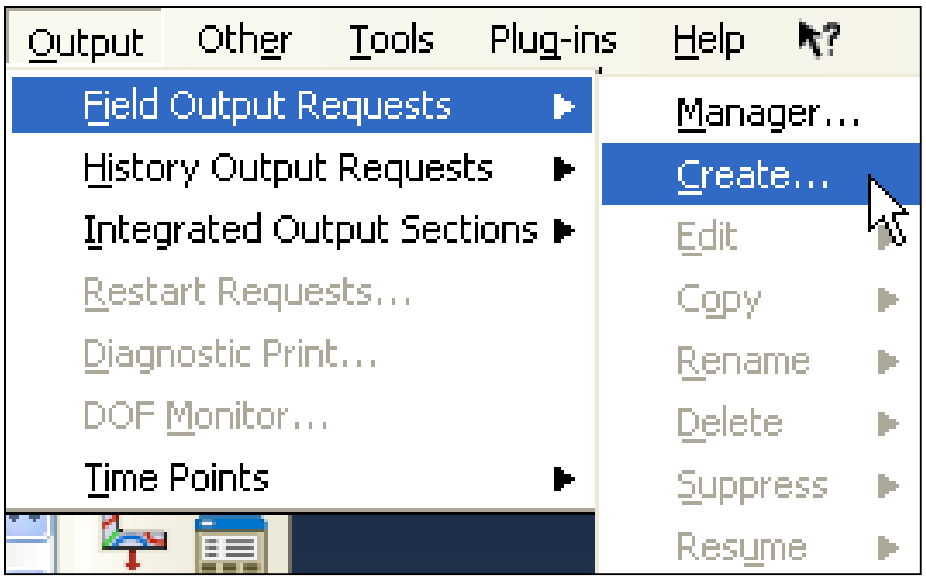
Analysis Steps and Procedures

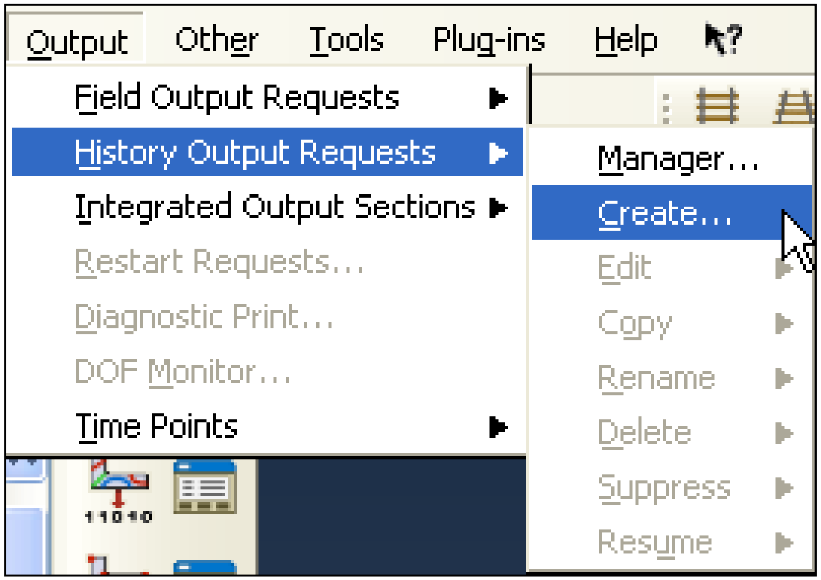
▶ Defining Steps in ABAQUS/CAE



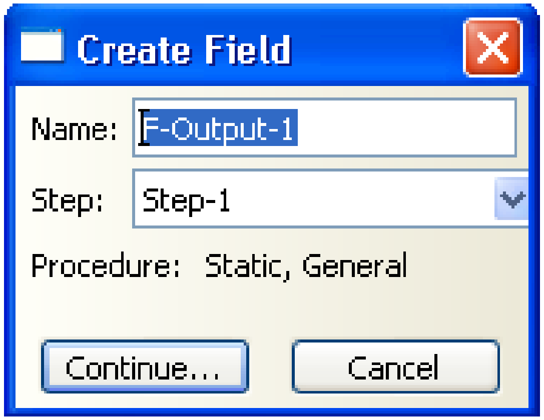
Output

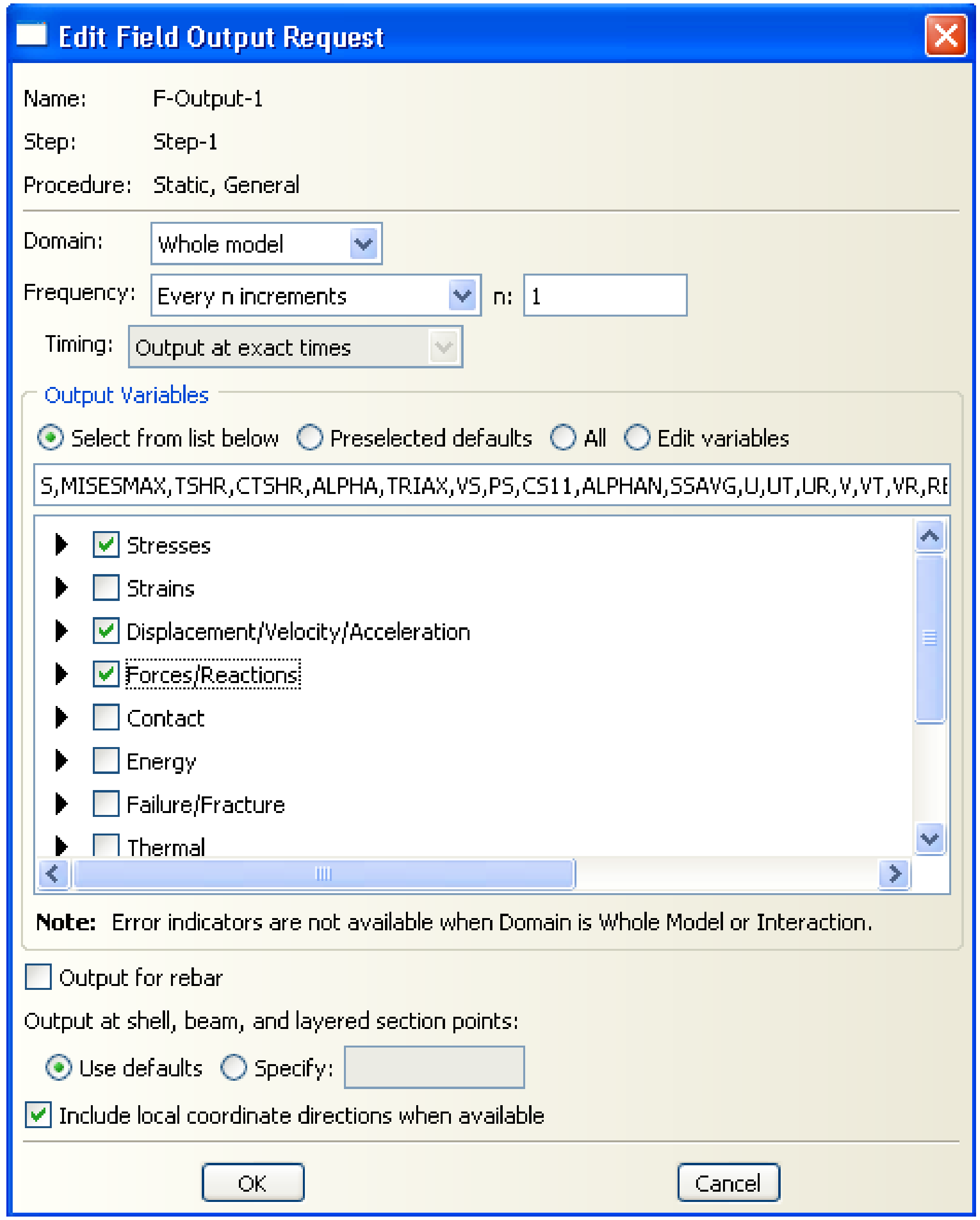
- **Two types of output data: field and history data**
  - Field data is used for model (deformed, contour, etc.) and X-Y plots.
  - History data is used for X-Y plots.



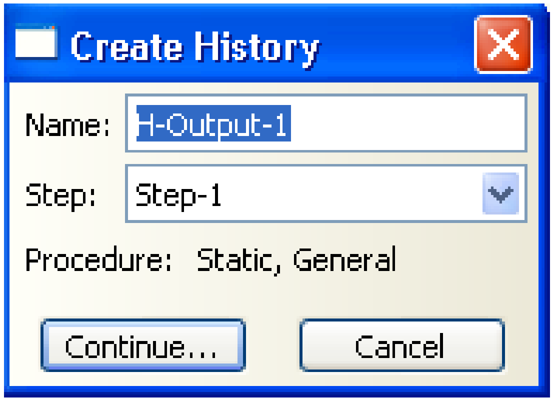


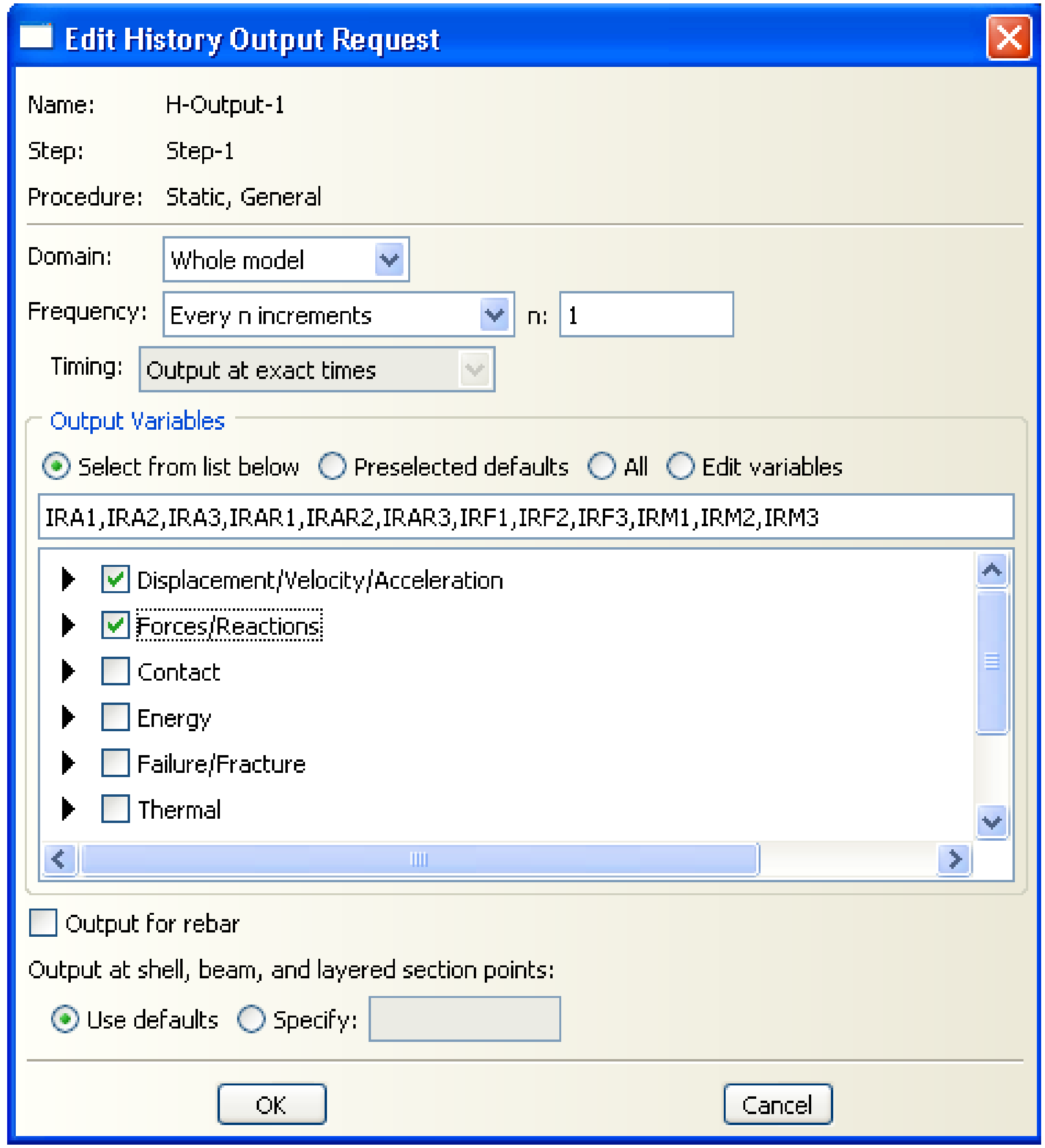
Field Output





History Output





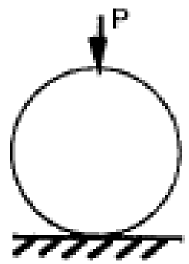
Loads and Boundary Conditions

- **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)

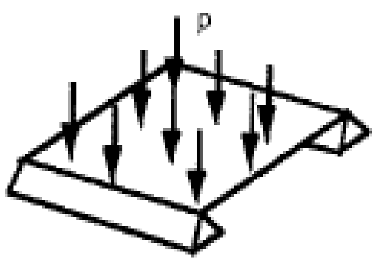
- **Sometimes the term refers specially to force-type quantities (as in the Load Manager of the Load module); for example,**
  - Concentrated forces, pressures, and body loads but not boundary conditions or temperature.

Loads and Boundary Conditions

Examples of loads



concentrated force

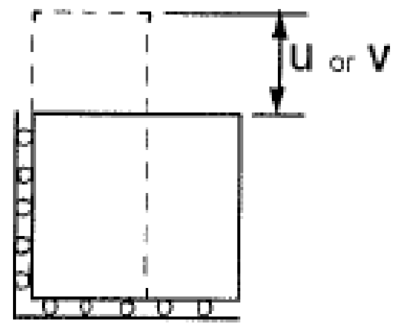


pressure load or
distributed heat flux

Examples of boundary conditions

fixed translations
and rotations

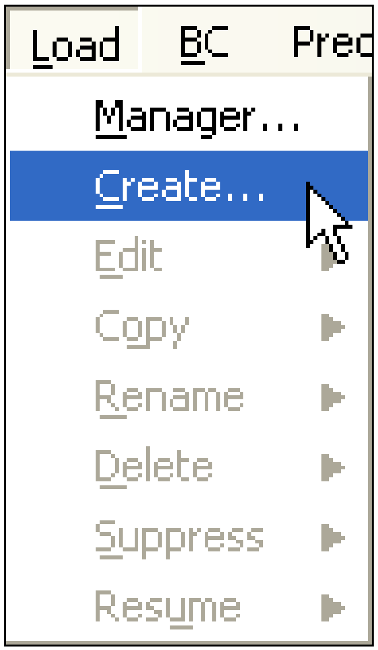
prescribed translations
or velocities

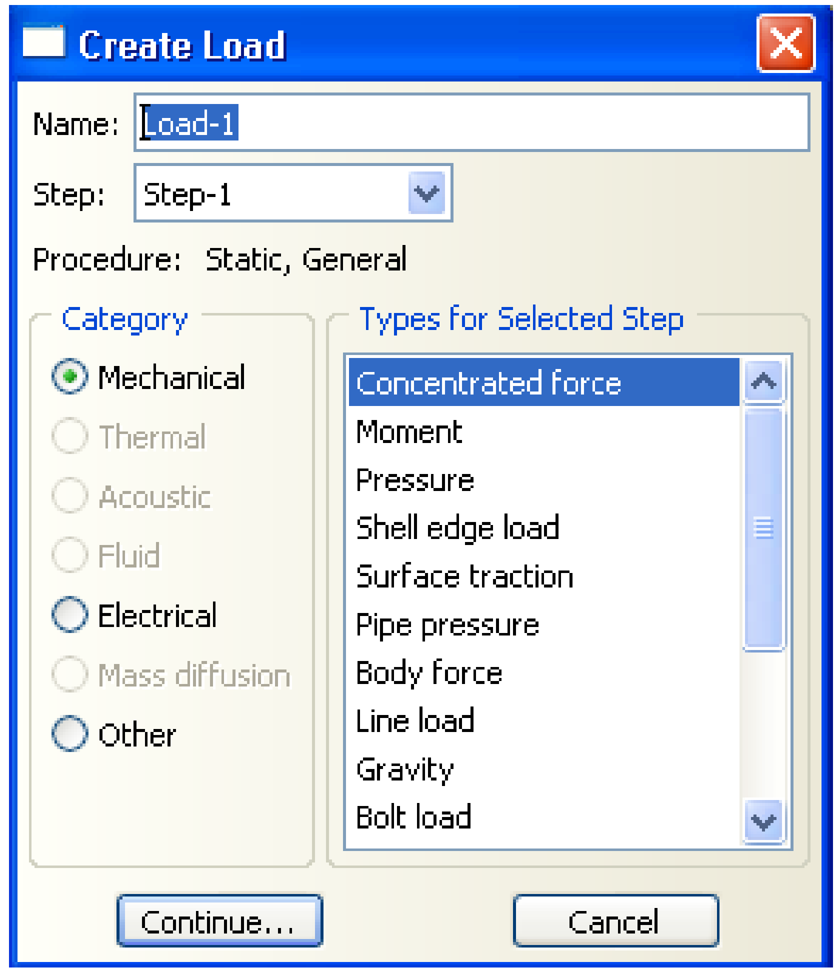


Loads

▶ Defining loads and boundary conditions in ABAQUS/CAE

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



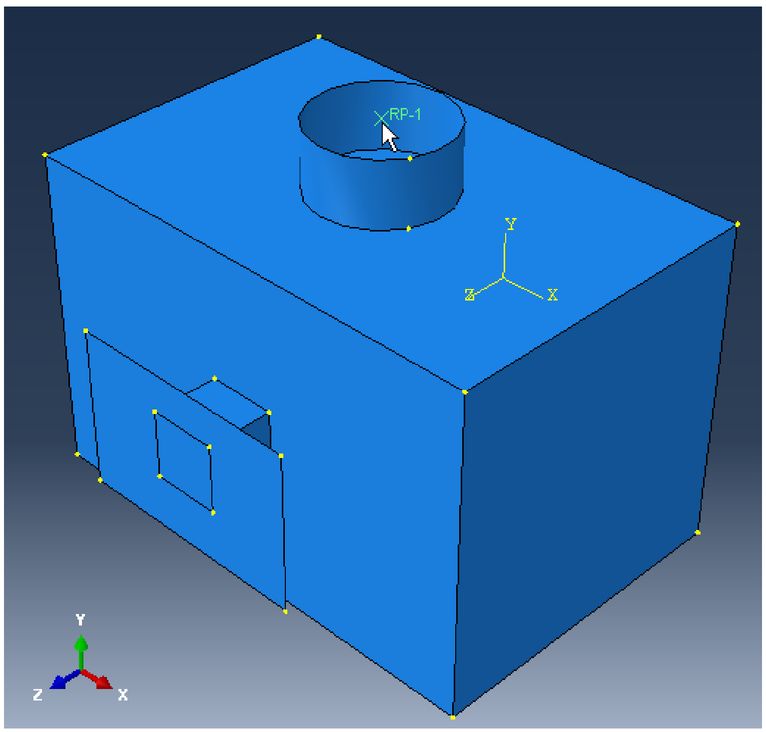


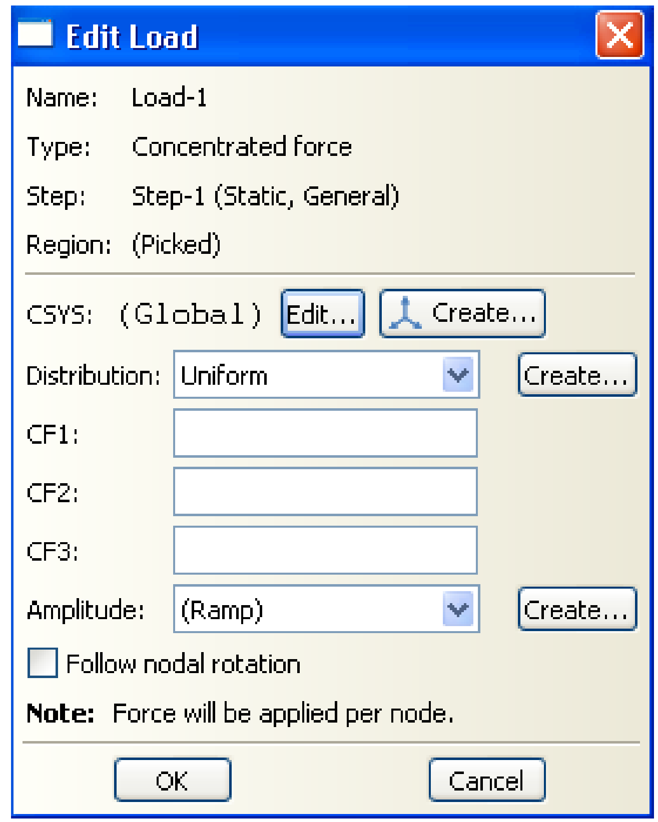
Loads

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

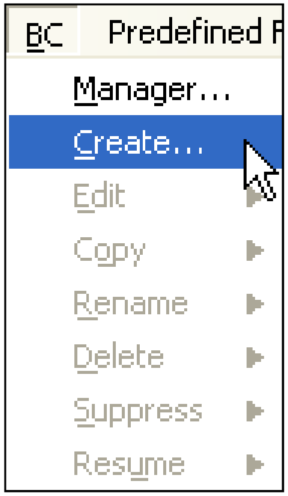
(Surfaces on orphan meshes can be selected by picking one element face and a feature angle. Individual “edits” make it easy to clean up anomalies.)

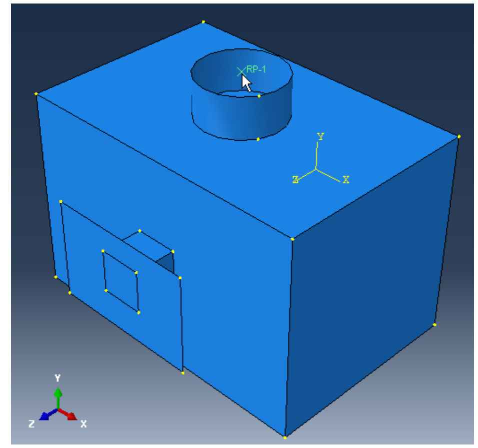
- Edit the load (or boundary condition) to specify magnitude, etc.

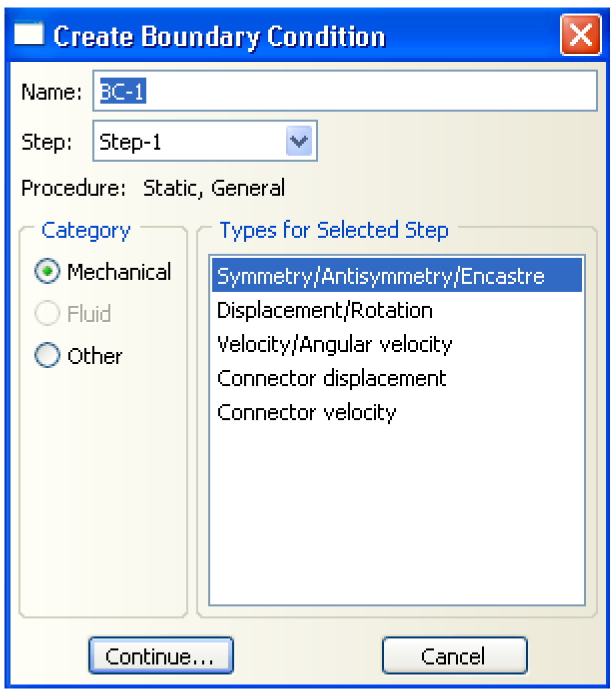


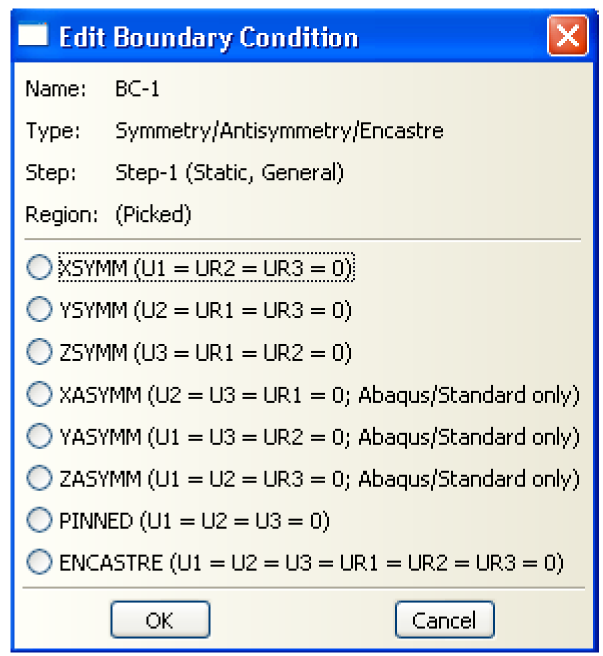


Boundary Conditions









Reference Point

CONTENTS

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

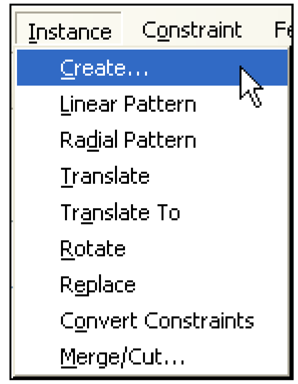
- **You can create either independent or dependent part instances**
  - Independent instances can be modified (e.g., they can be partitioned).
  - Dependent instances cannot be modified (e.g., they cannot be partitioned)
  - Dependent instances share the same geometry and mesh of the original part so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

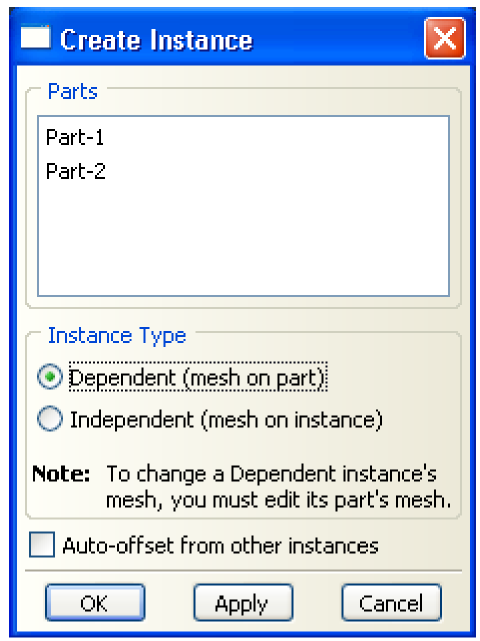
- **All instances of a part must be either dependent or independent**
  - No mixture is allowed for a given part

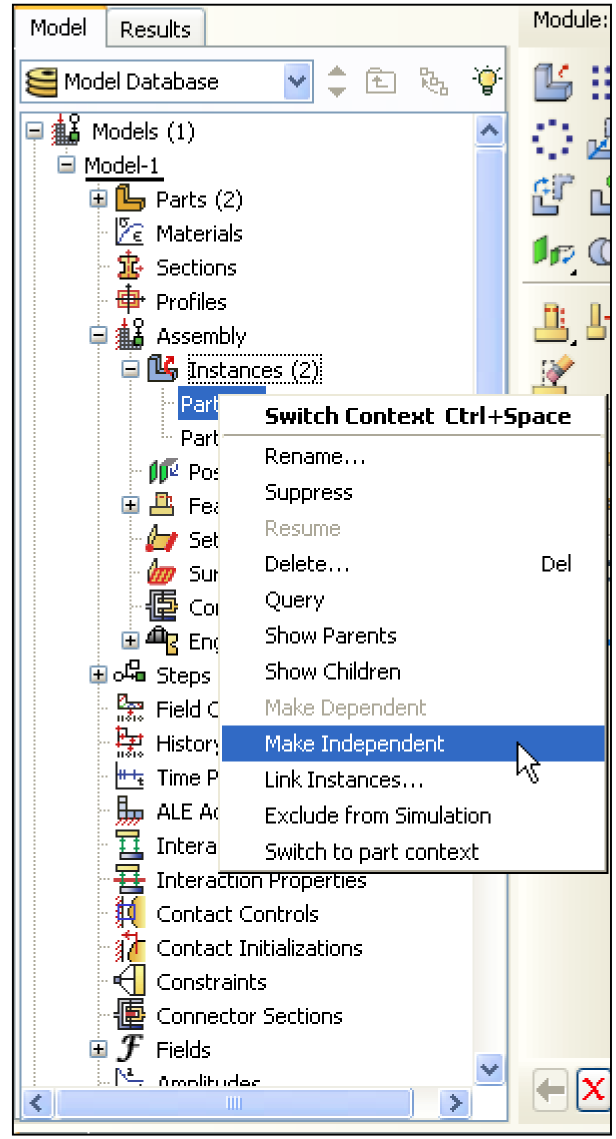
- **All orphan mesh instances must be dependent**

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**





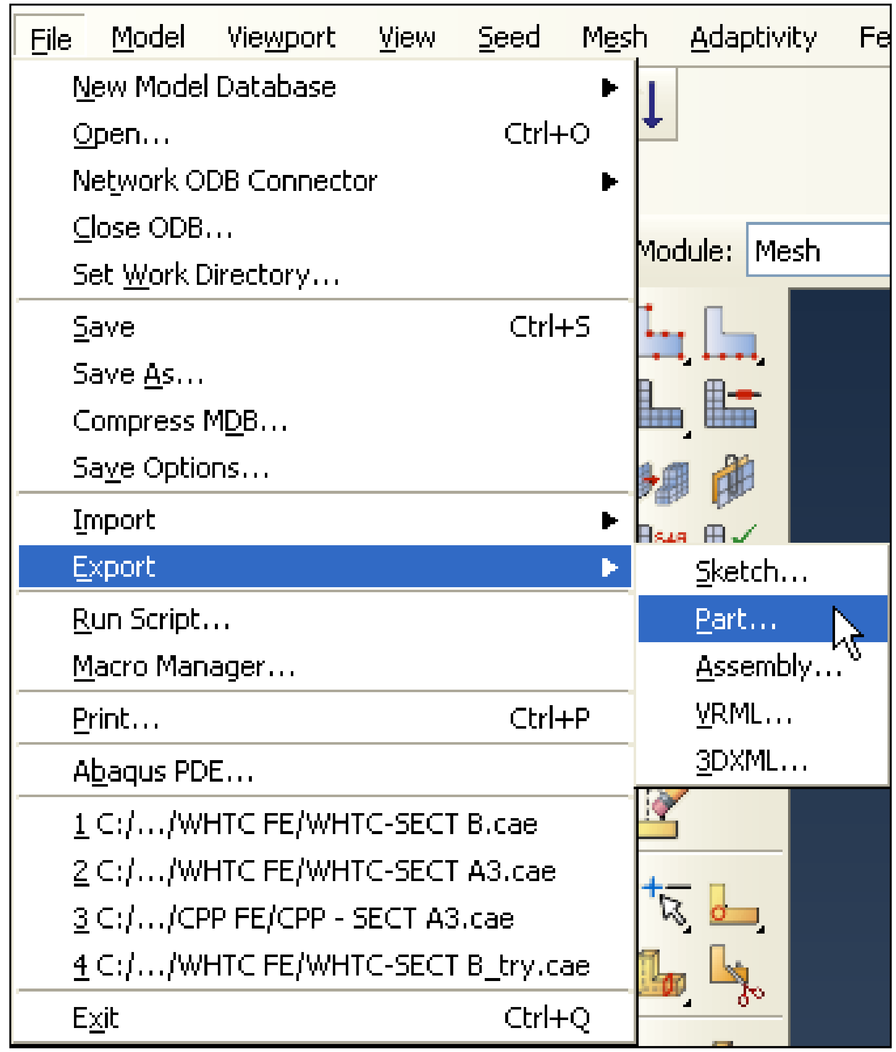


Exploring Mesh

- 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**



CHAPTER

II. Meshing Techniques

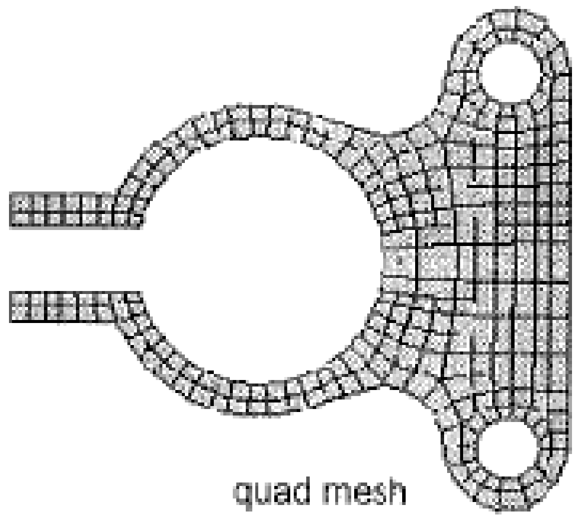
Meshing Techniques

► Free meshing

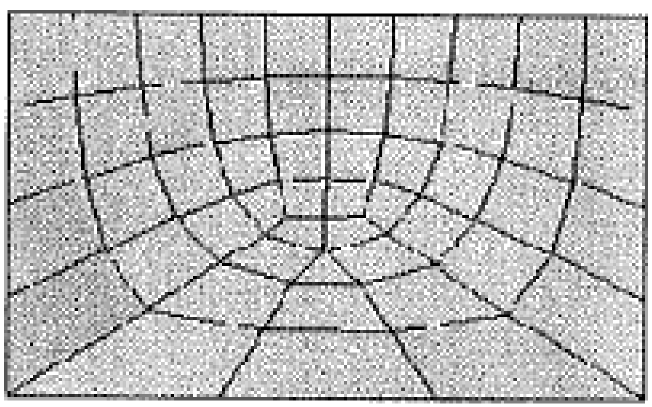
- Free meshing uses no pre-established mesh patterns, making it impossible to predict a free mesh pattern before creating mesh
- Element shape options available for free meshing two-dimensional regions:

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

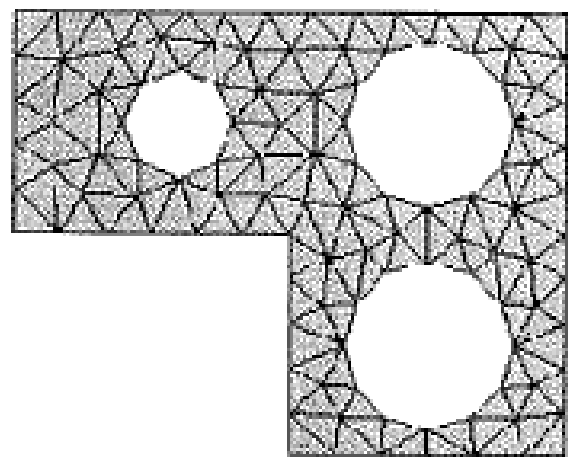
Meshing Techniques



quad mesh



quad-dominated mesh

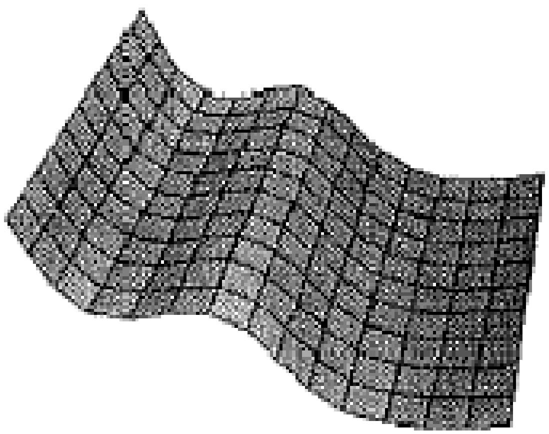


triangular mesh

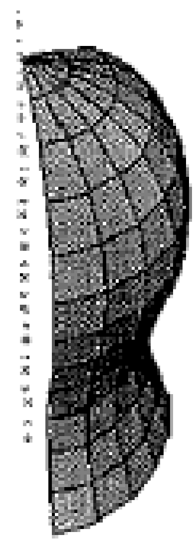
Meshing Techniques

► 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



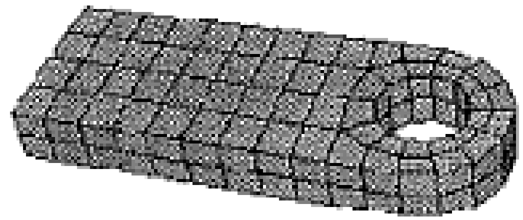
Degenerate revolved mesh

Meshing Techniques

► Swept Meshing

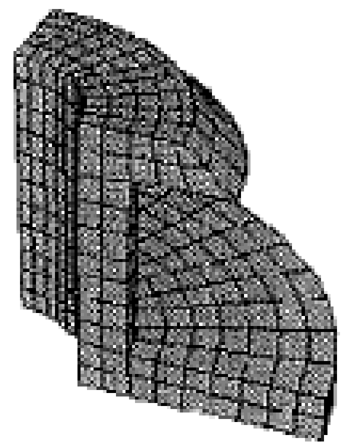
- Swept solid regions can be filled with:
  - Hex meshes
  - Hex-dominated meshes
  - Wedge meshes
- The swept path can be:
  - Straight line
  - Circular arc
  - Spline

sweep path: straight line



extruded mesh

sweep path: arc

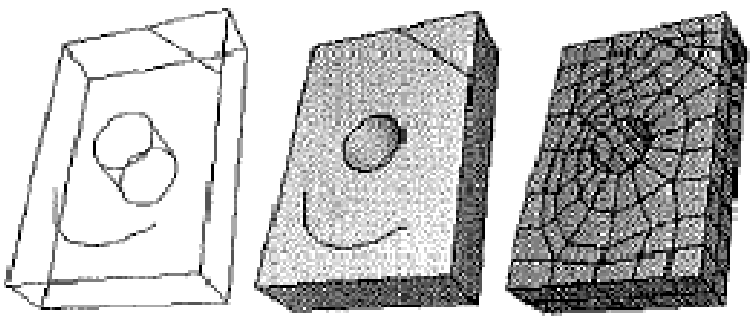


revolved mesh

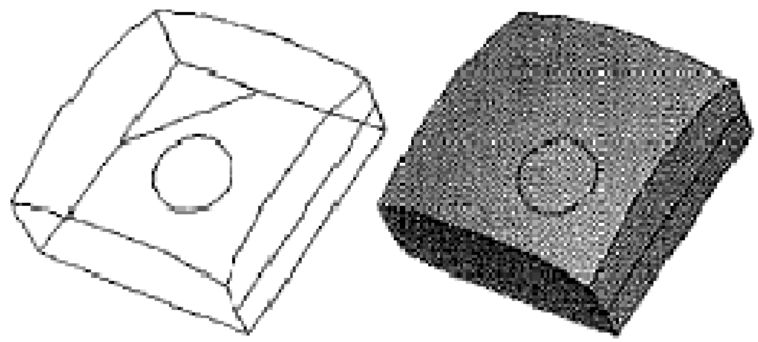
Meshing Techniques

► 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

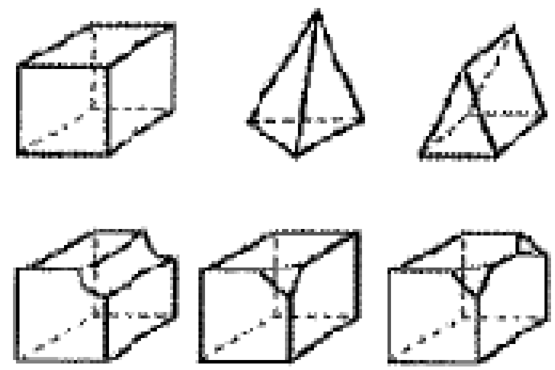


Not swept meshable

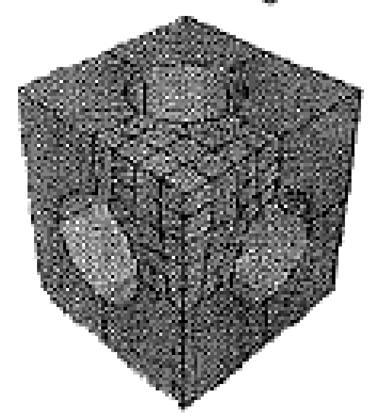
Meshing Techniques

► 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.



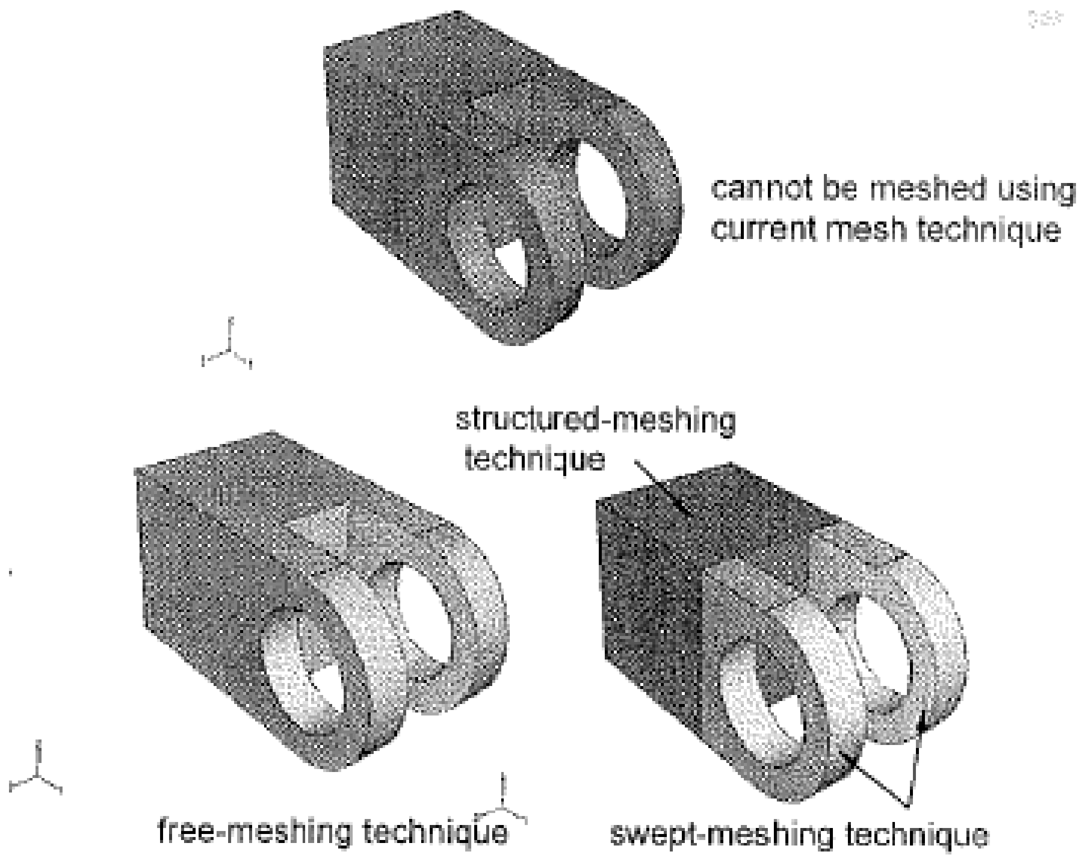
three-dimensional structured meshable regions



simple mesh topology

Meshing Techniques

- **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

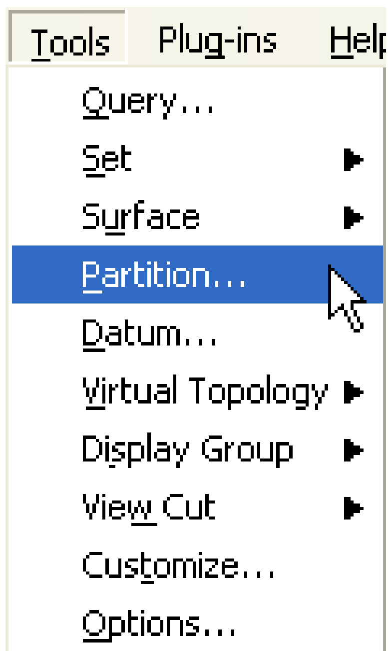


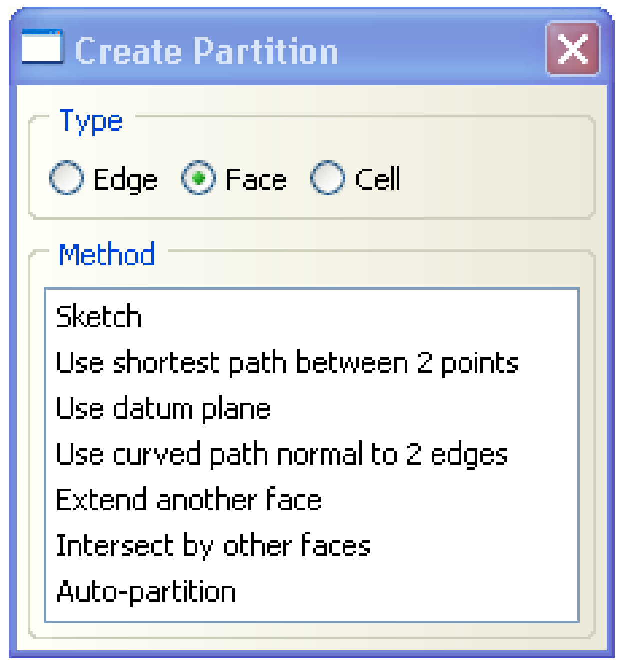
Meshing Techniques

► 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.

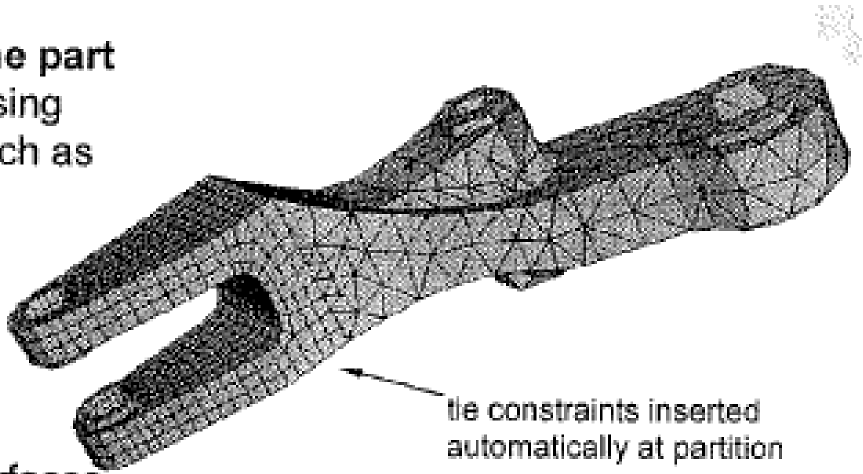
- Partitioning can be used to:
  - Change and simplify the topology so that the regions can be meshed using primarily hexahedral elements with the structured or swept meshing techniques.





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.



Mesh Compatibility

– Currently it is not possible to automatically obtain meshes that are compatible between part instances

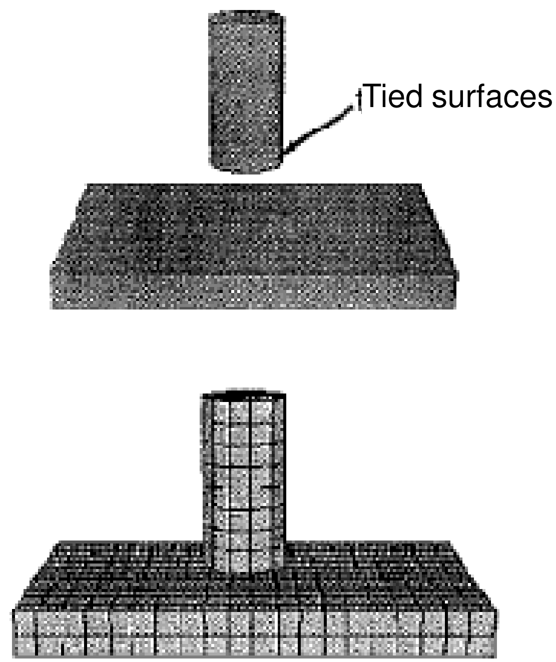
– If mesh compatibility is required between two or more bodies, first try to create a single part that contains all the bodies.

  - 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.

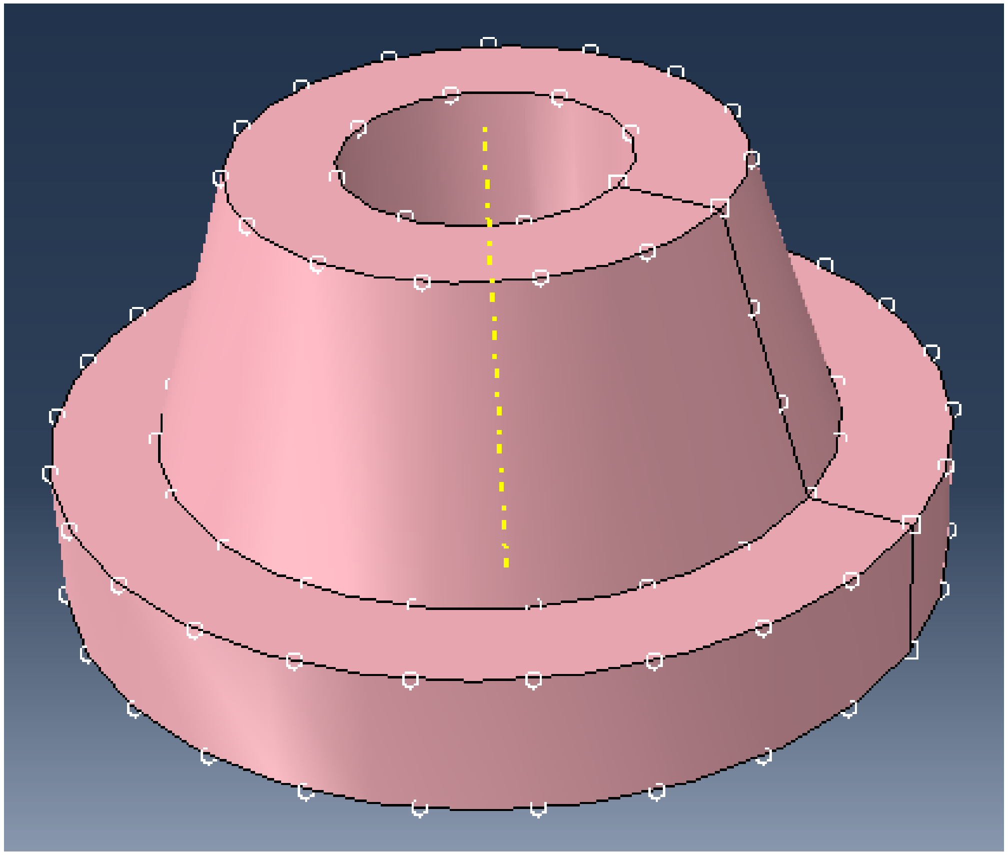


Using tie constraints to glue the cylinder to the block: exploded view of assembly (top) and mesh

Controlling Mesh Density and Gradation

► Mesh seeds

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

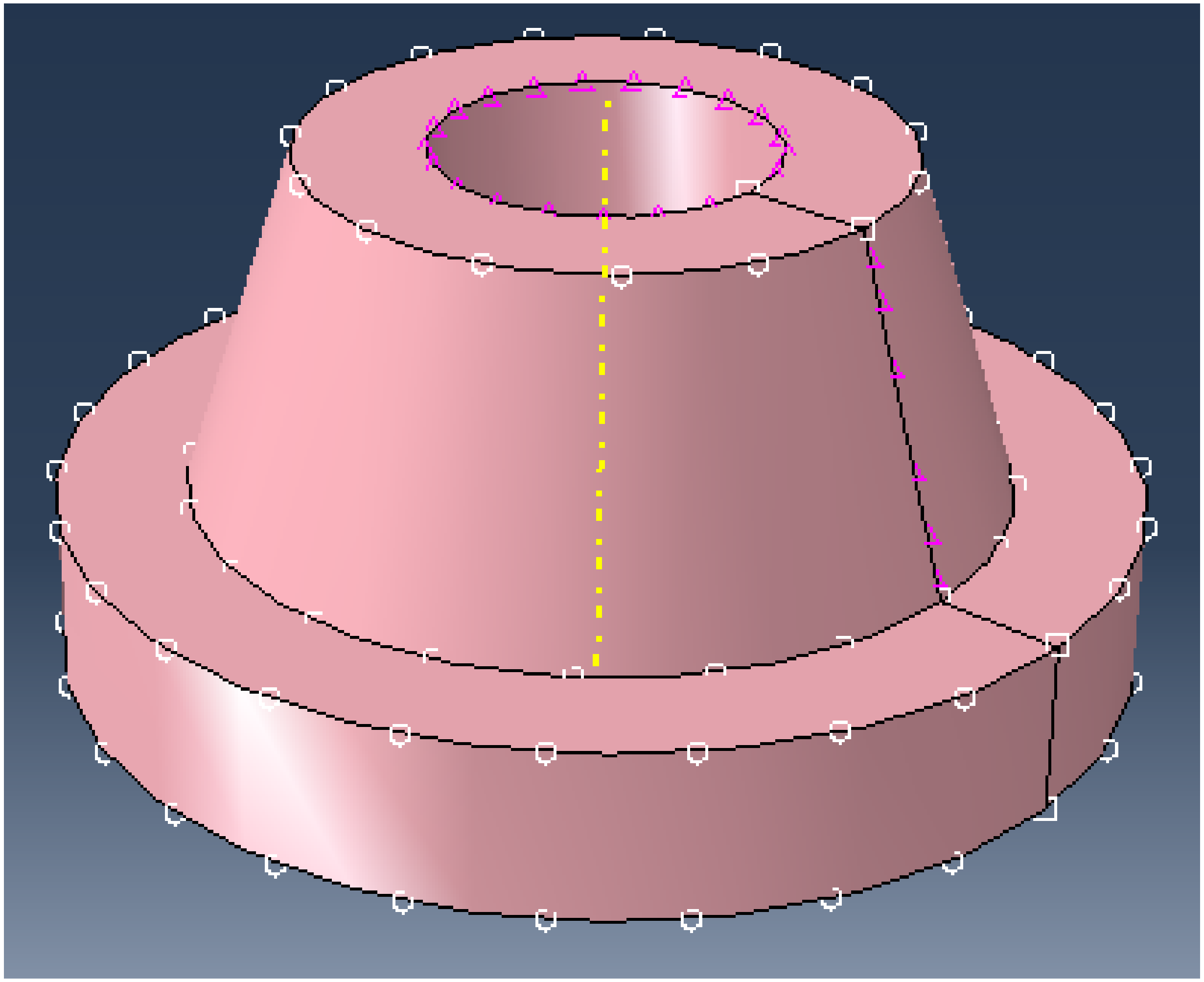


Controlling Mesh Density and Gradation

- **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.

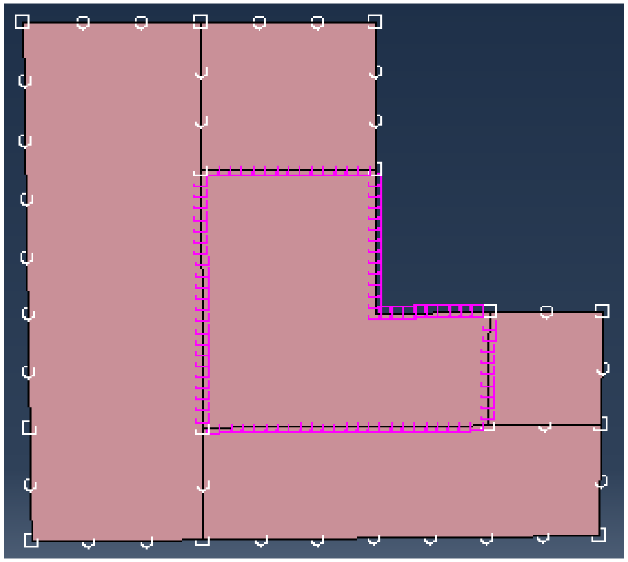
Controlling Mesh Density and Gradation

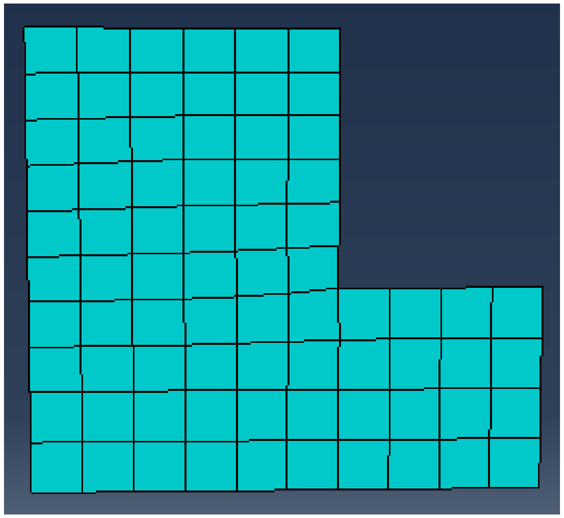
- **Global seeds (black) and local seeds (magenta)**

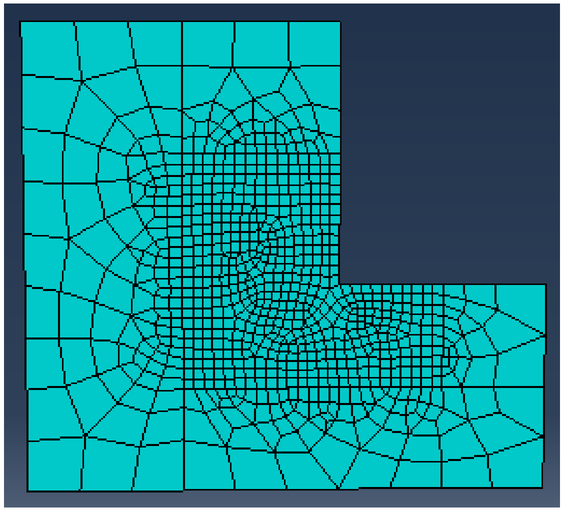


Controlling Mesh Density and Gradation

- **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.



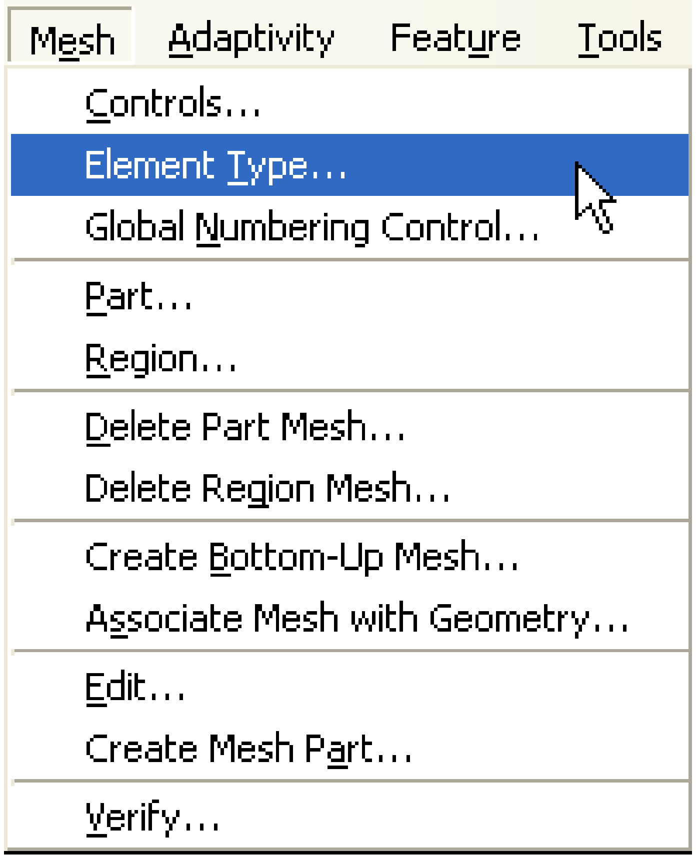




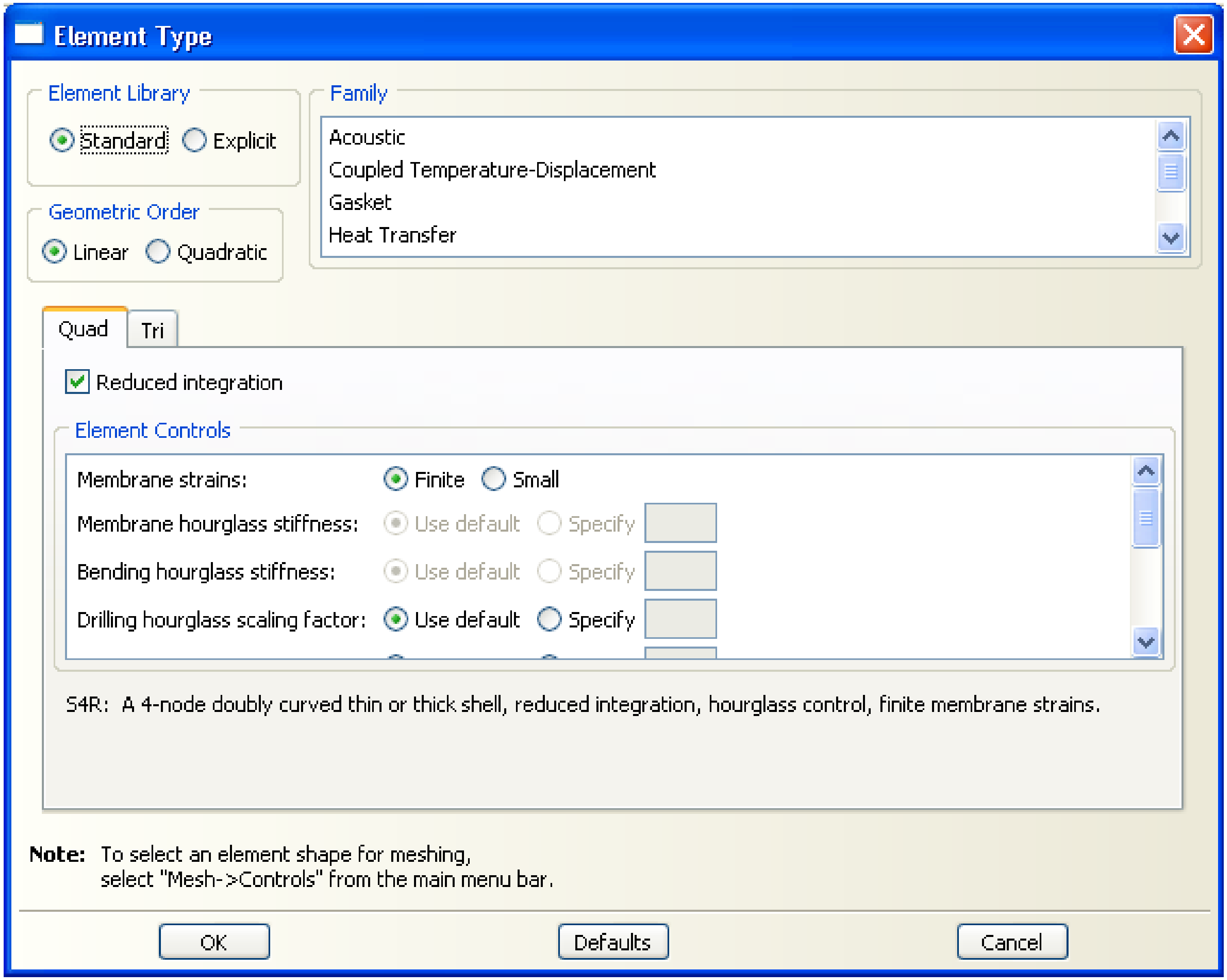
Partitioning and local mesh seeding allows you to refine mesh in the area of a stress concentration.

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

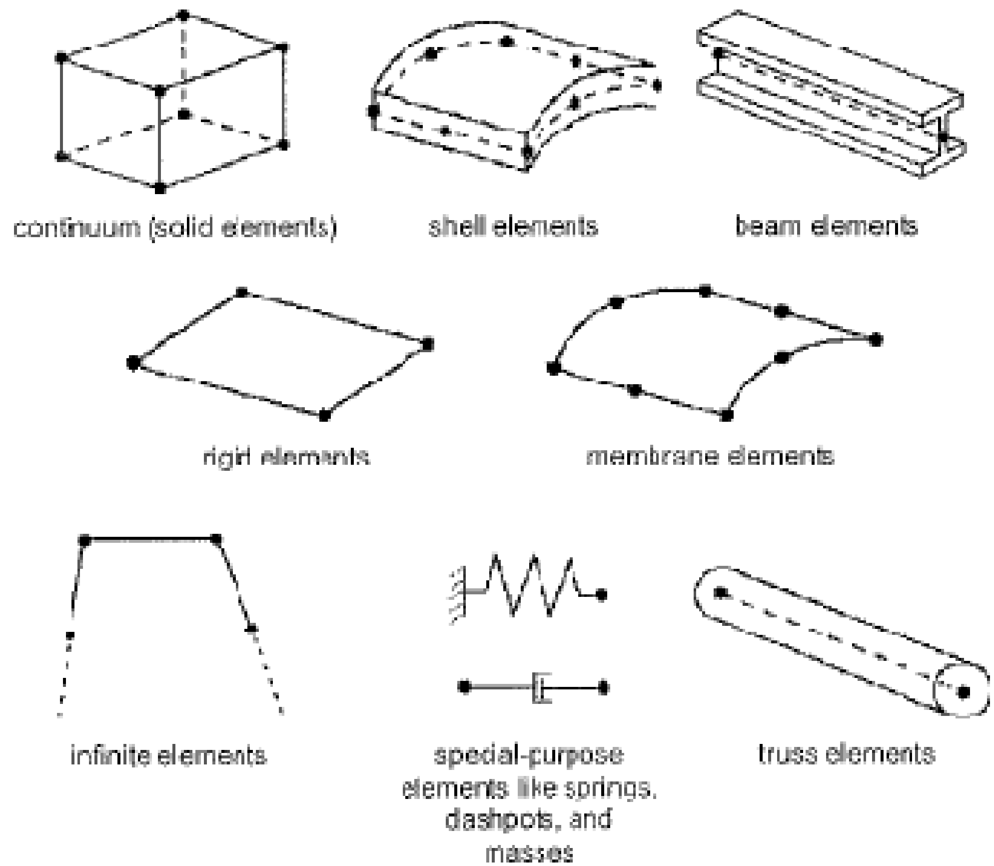


Assigning Element Types



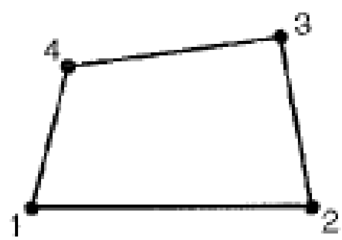
Element in ABAQUS

- **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.



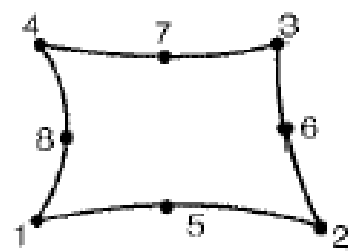
Element in ABAQUS

- **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

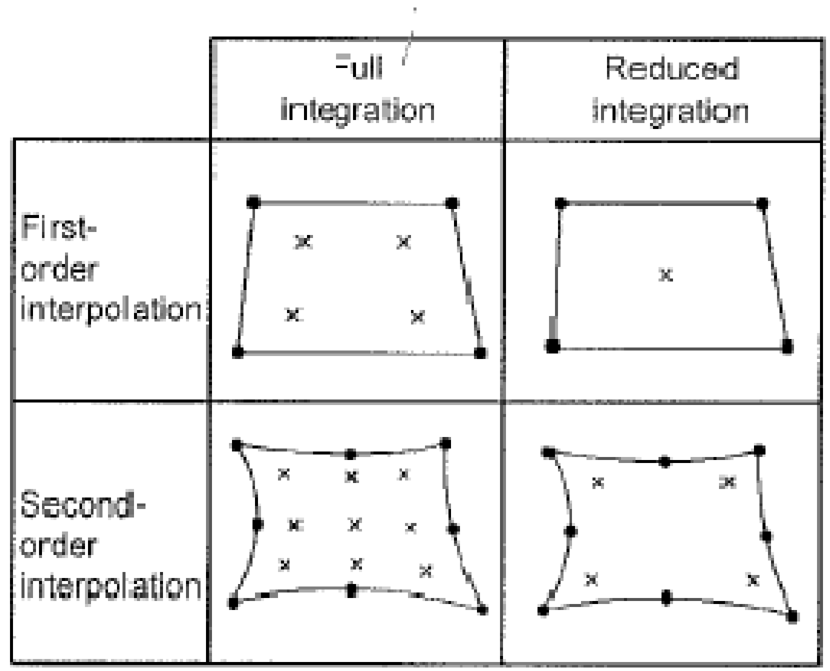
- The stiffness and mass of an element are calculated numerically at sampling points called “integration points: within the element
- The numerical algorithm used to integrate these variables influences how an element behaves

▶ 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

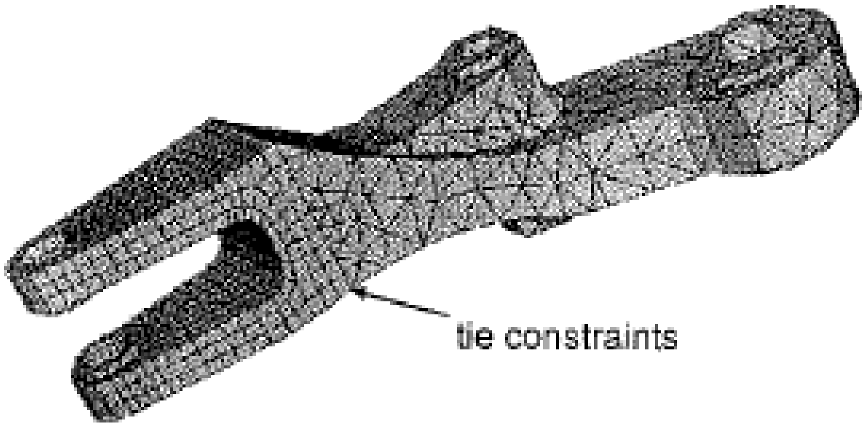
Interaction - Constraints

What are constraints?

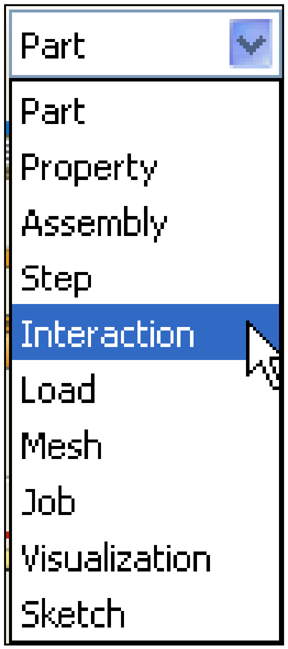
- Constraints allow you to model kinematic relationship between points

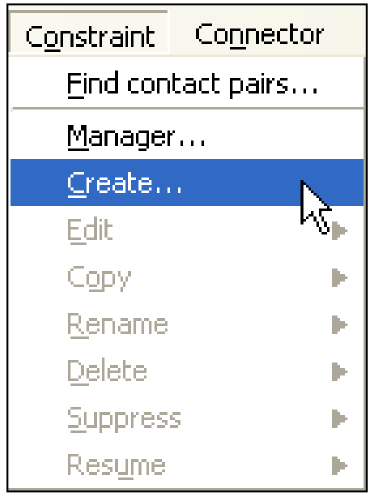
Tie constraints

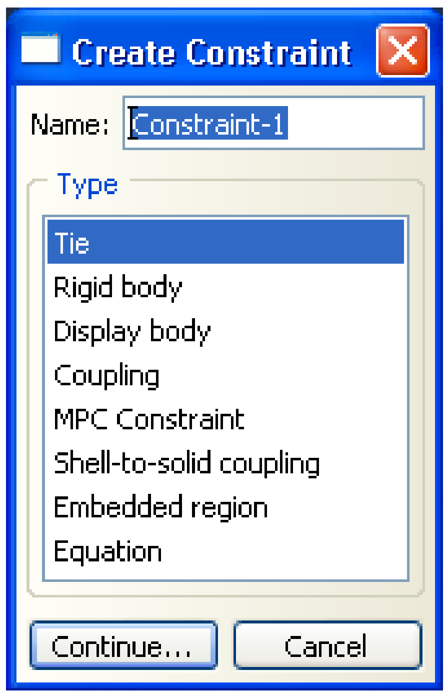
- Allow you to fuse together two regions even though the meshes created on the surfaces of the regions may be dissimilar
  - Easy mesh transitioning
- It provides a simple way to bond surfaces together permanently
- Surface-based constraint using a master-slave formulation
- The constraint prevents slave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

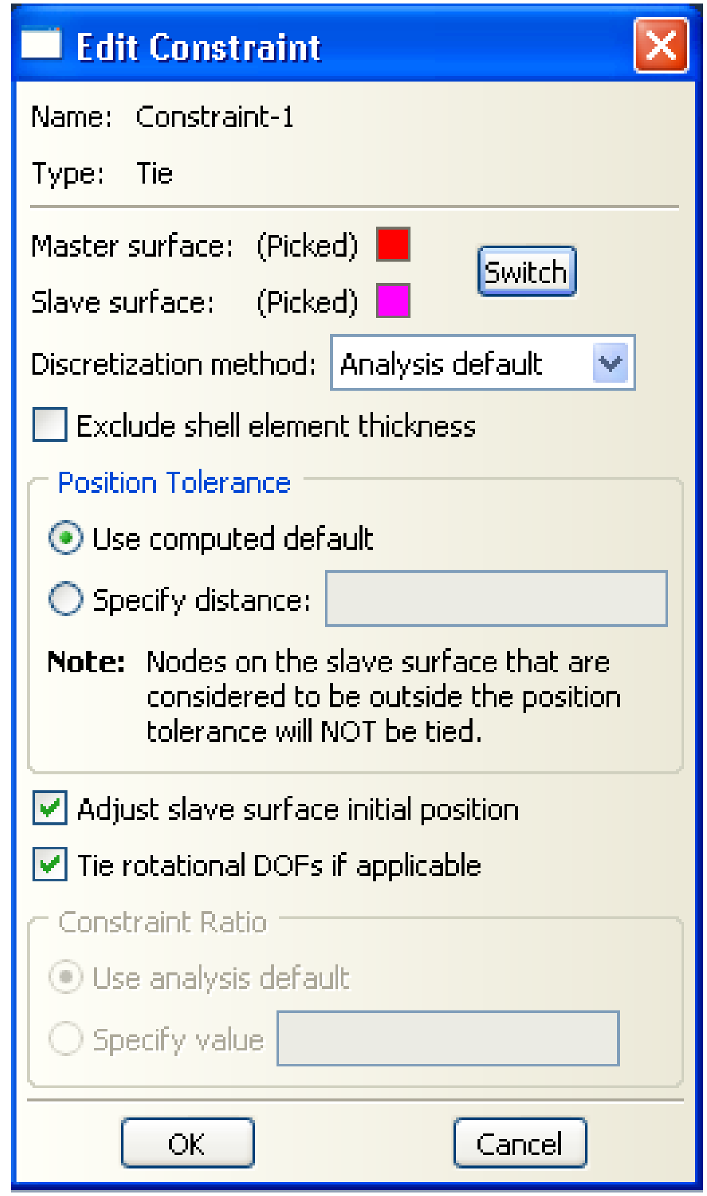


Constraints - Tie

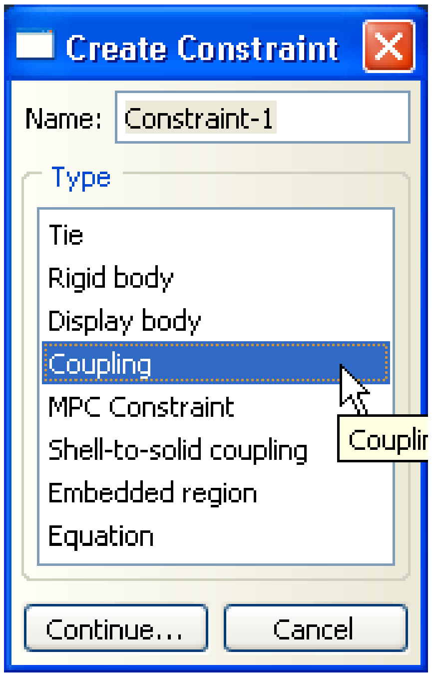


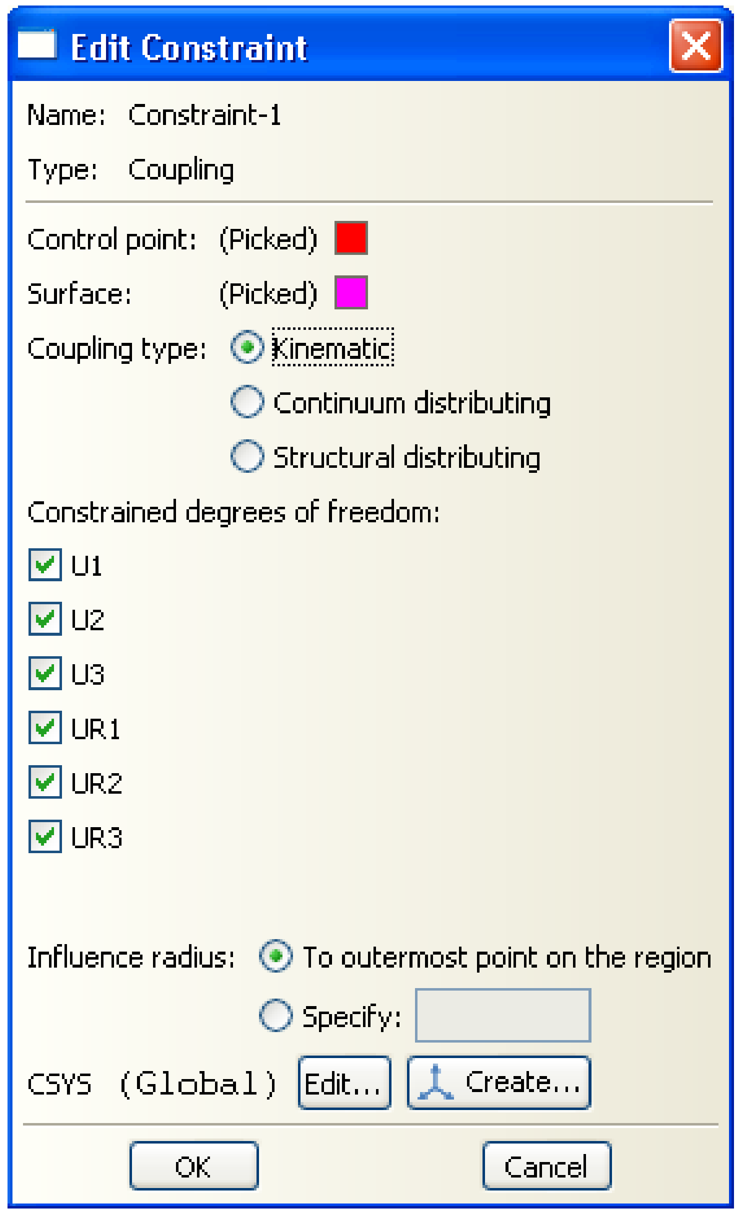


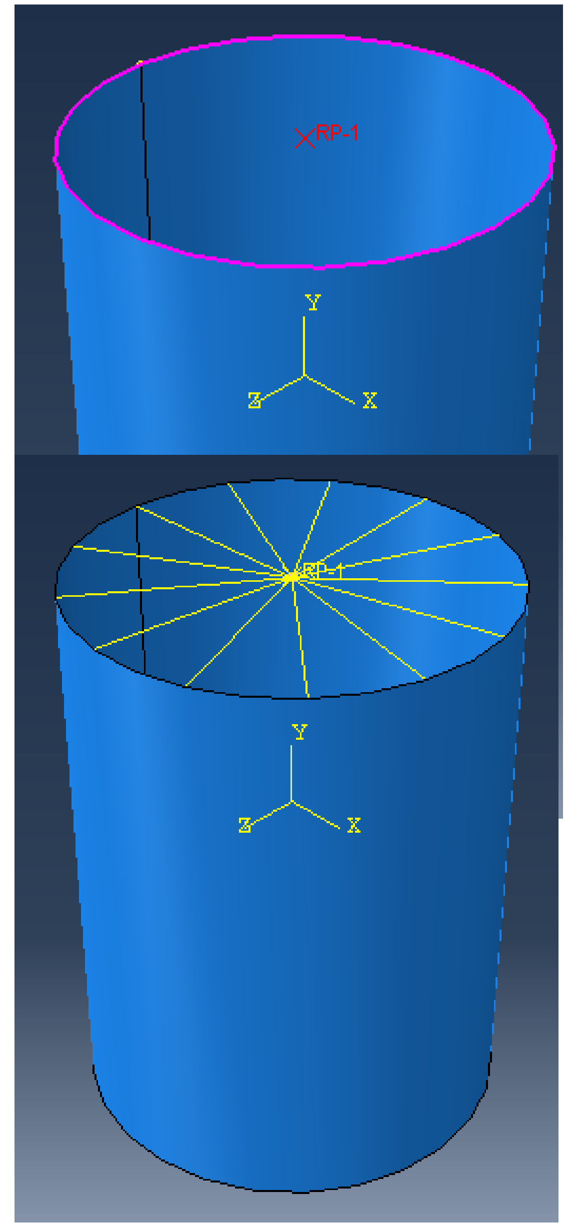




Constraints - Coupling







Session 4

I. Job Management and Results Visualization

II. Stress Concentration Factors (SCF)

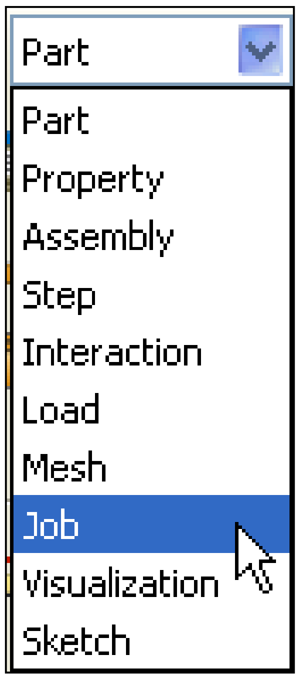
III. Common Error Messages

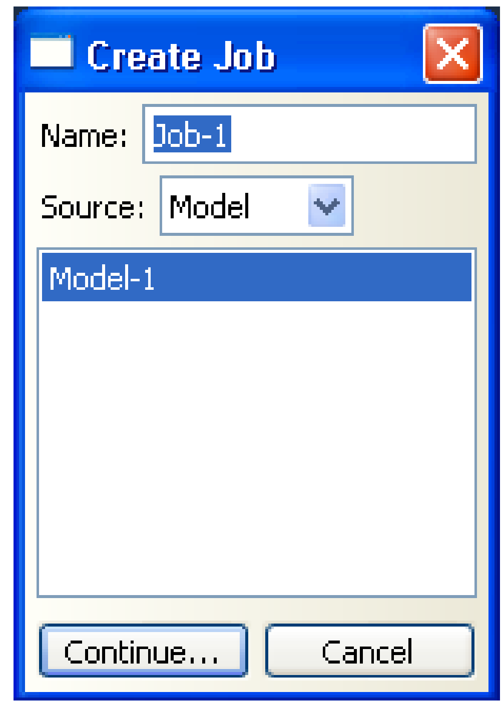
IV. Tutorial 1 – Linear Static Analysis

CHAPTER

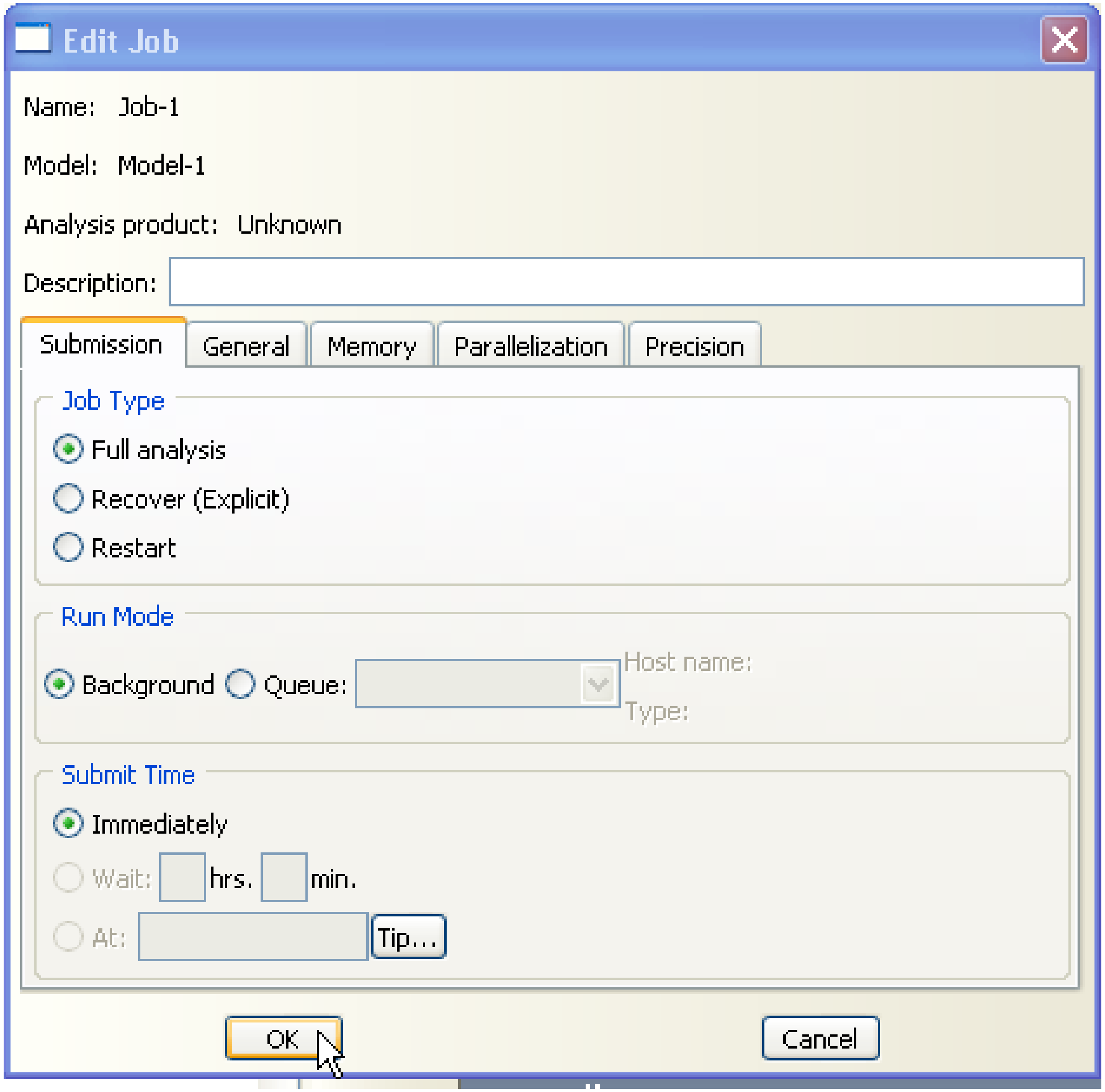
I. Job Management and Results Visualization

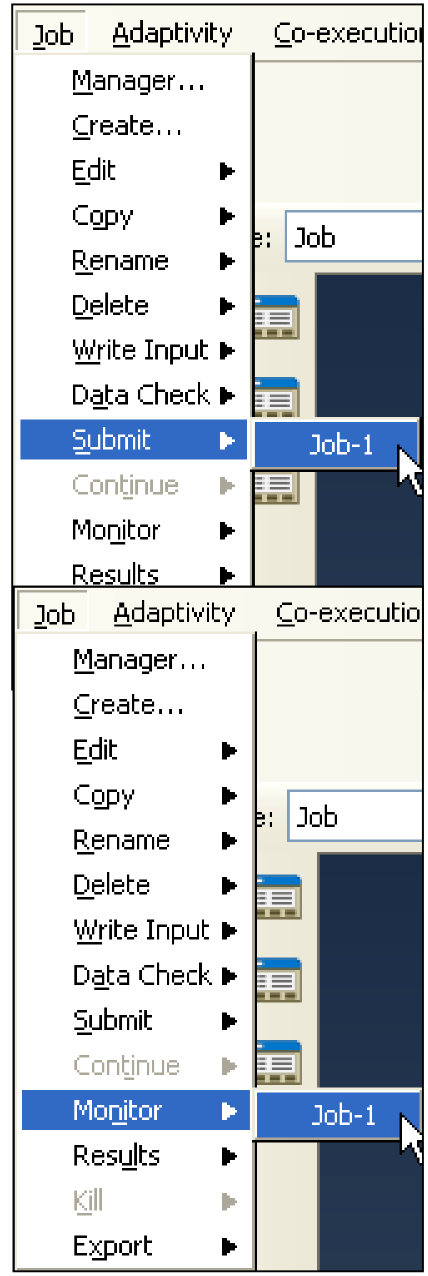
Job Management



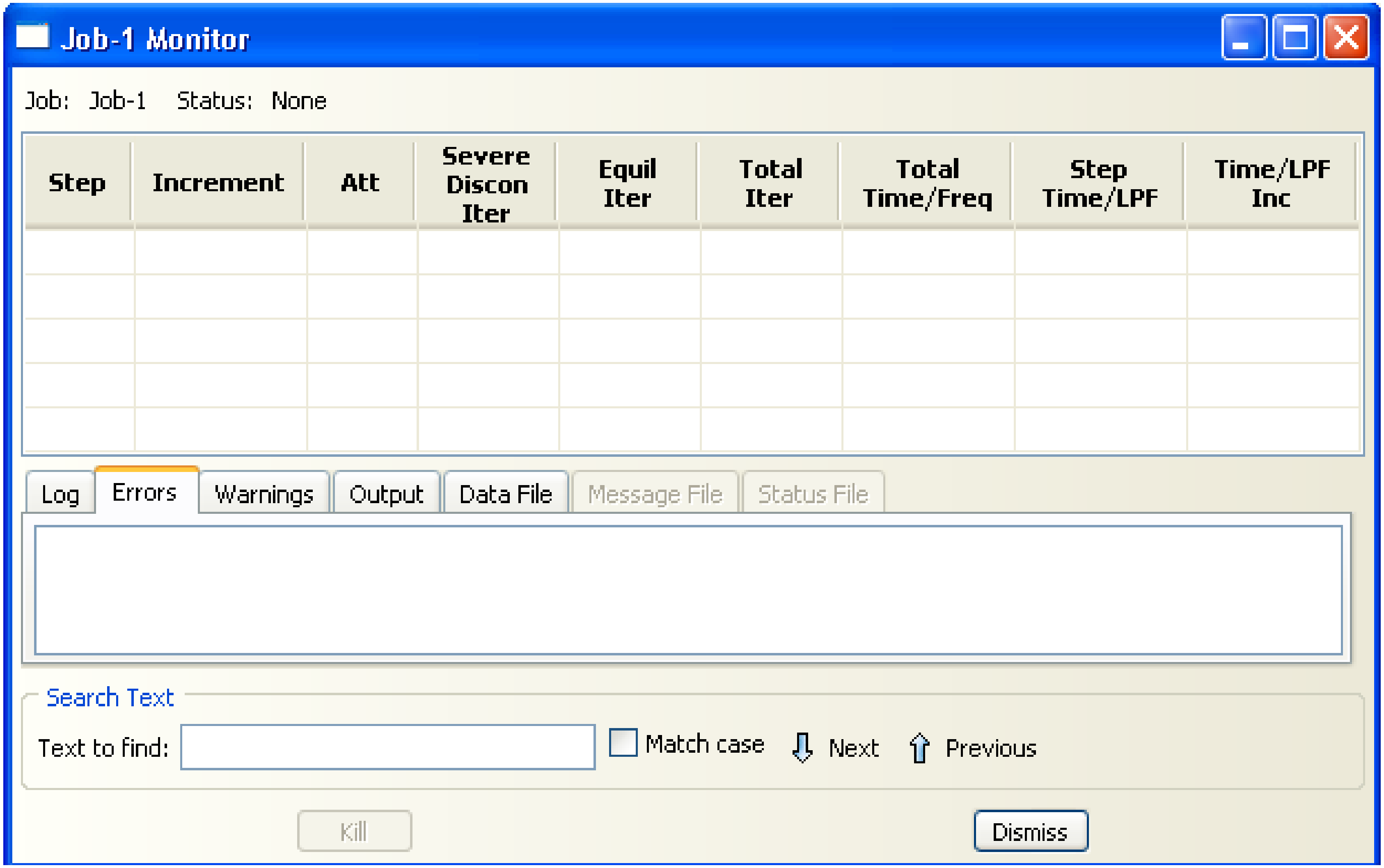


Job Management

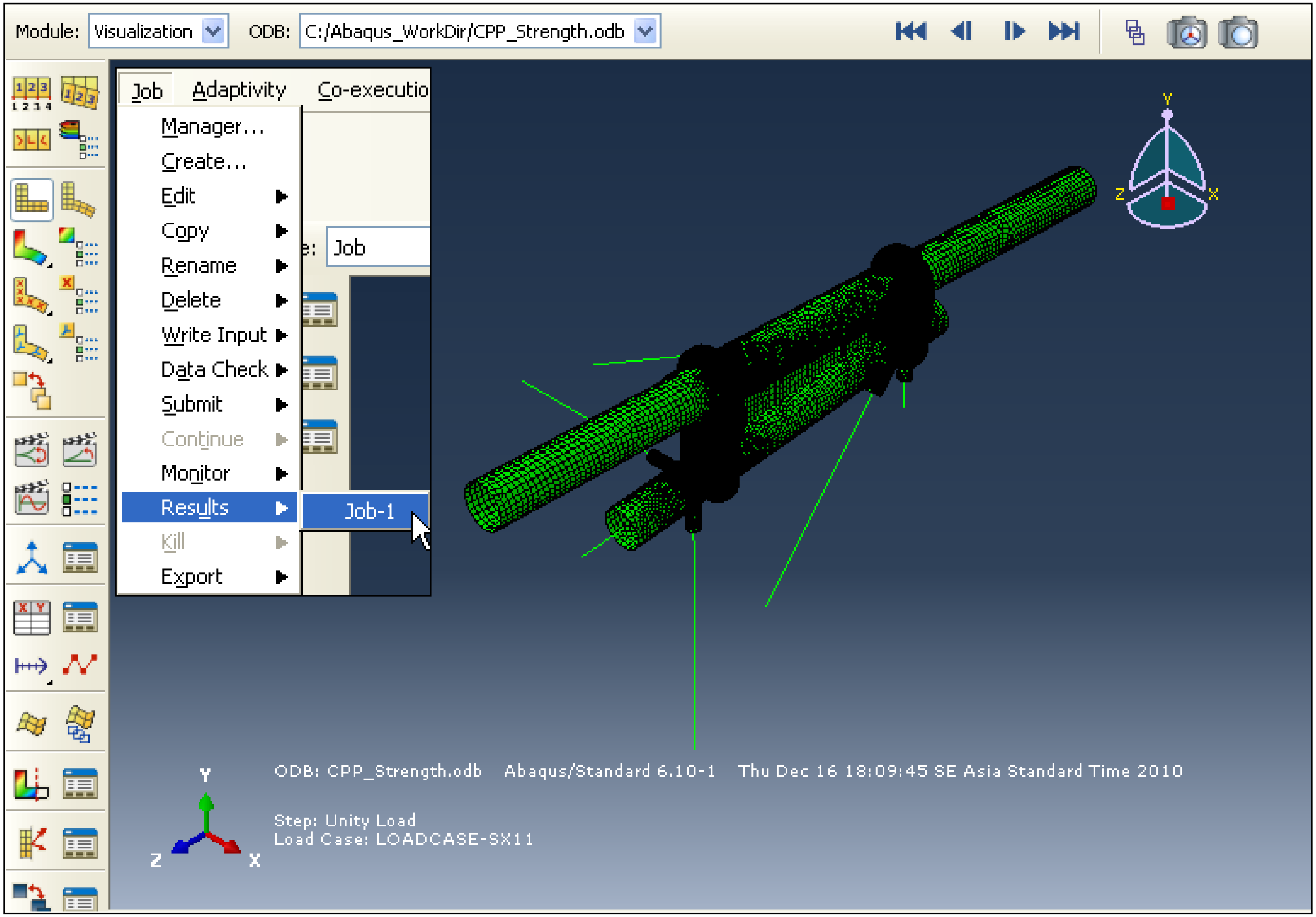




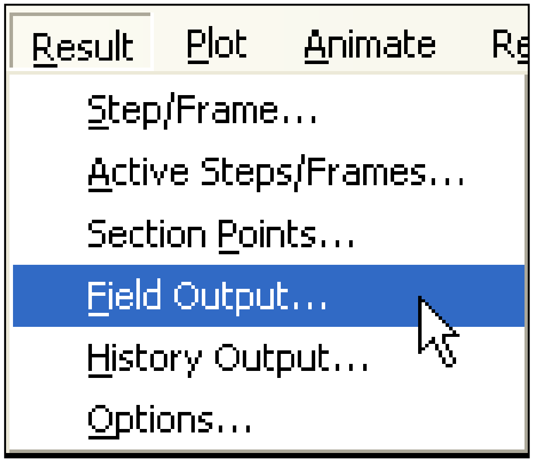
Job Management

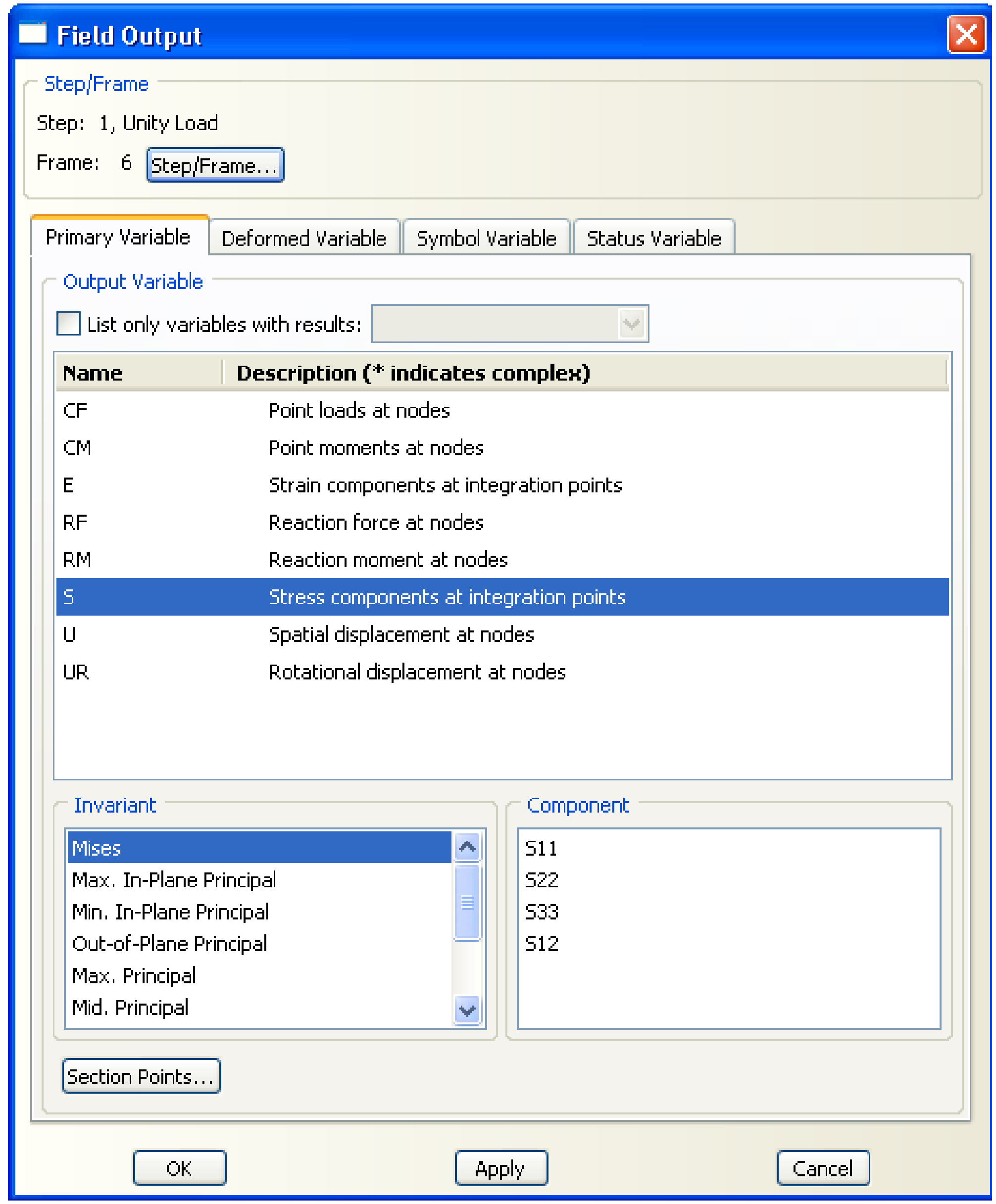


Results Visualization

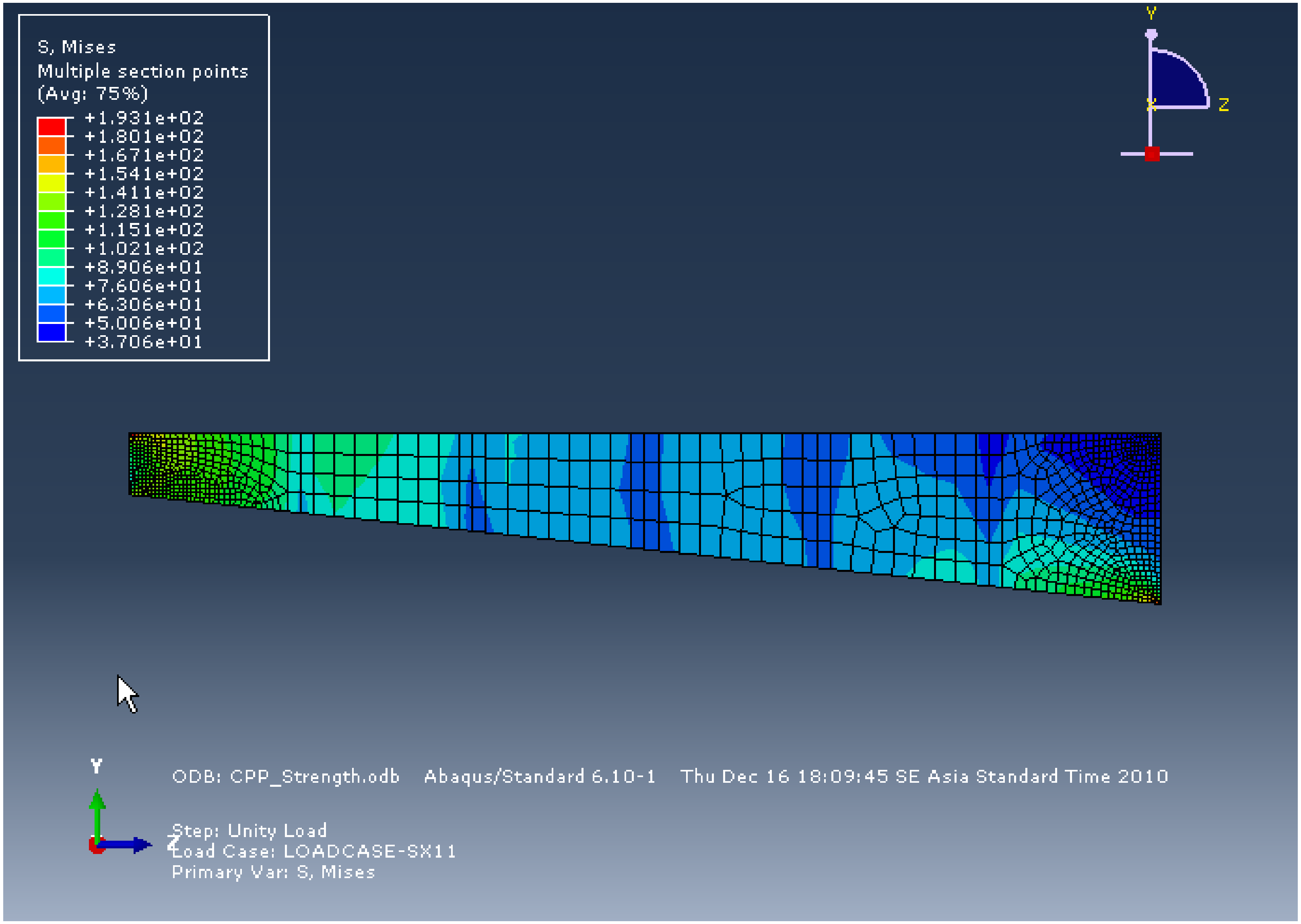


Results Visualization





Results Visualization

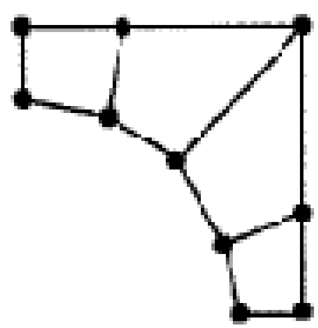


CHAPTER

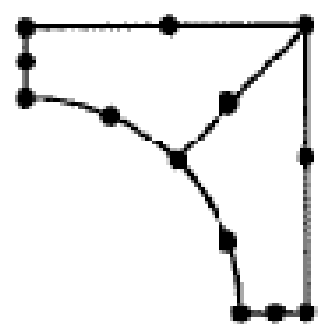
II. Stress Concentration Factors (SCF)

Stuss 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-order elements—element faces are straight line segments



Model with second-order elements—element faces are quadratic curves

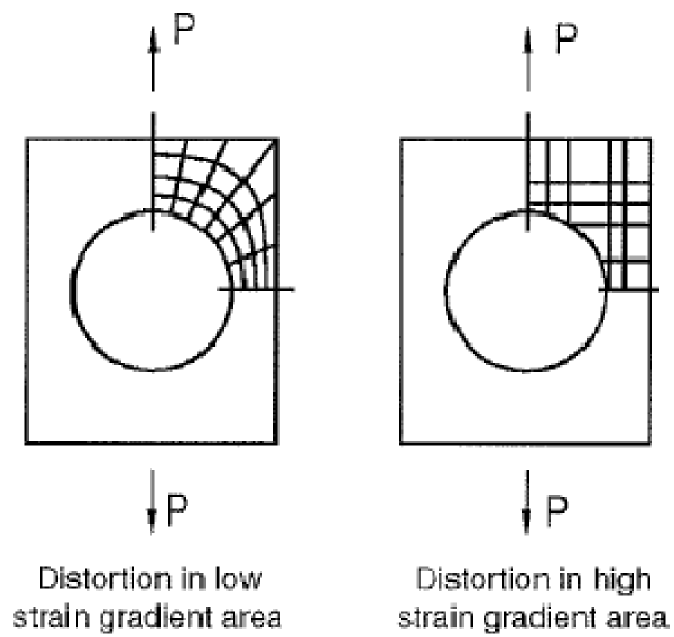
Stress Concentration

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

Element type	σ_{yy} at D (Target=100.0)	
	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

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