Lecture 2

Elements

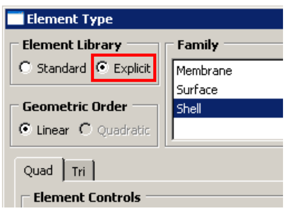
Overview

- Introduction
- Solid Elements
- Shell and Membrane Elements
- Beam and Truss Elements
- Rigid bodies
- Special-Purpose Elements
- Hourglassing
- Second-order Accuracy

Introduction

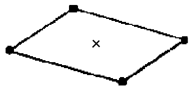
Introduction

- **The wide range of elements in the ABAQUS/Explicit element library provides flexibility in modeling different geometries and structures.**
  - Each element can be characterized by considering the following:
    - Family
      - Continuum, shell, membrane, rigid, beam, truss elements, etc.
    - Number of nodes
      - Element shape
      - Geometric order
        - Linear or quadratic interpolation
    - Degrees of freedom
      - Displacements, rotations, temperature
    - Formulation
      - Small- and finite-strain shells, etc.
    - Integration
      - Reduced and full integration

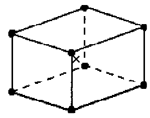


Introduction

- **Each element in ABAQUS has a unique name, such as S4R, B31, M3D4R, C3D8R and C3D4.**
  - The element name identifies primary element characteristics.

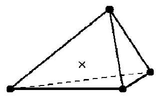


S4R: **S**hell, **4**-node, **R**educed integration

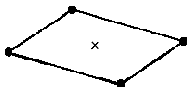


C3D8R: **C**ontinuum, **3-D**, **8**-node, **R**educed integration

B31: **B**eam, **3**-D, **1**st-order interpolation



C3D4: **C**ontinuum, **3-D**, **4**-node



M3D4R: **M**embrane, **3-D**, **4**-node, **R**educed integration

Introduction

- **General characteristics of the ABAQUS/Explicit element library**
  - ABAQUS/Explicit (like other explicit codes) uses the lumped mass formulation for all elements.
  - ABAQUS/Explicit (like other explicit codes) uses reduced integration elements.
    - Reduced integration elements are computationally inexpensive.
      - With explicit methods, the performance bottleneck tends to be the element computations.
    - Exceptions: fully-integrated membrane elements, triangular and tetrahedral elements.
  - ABAQUS/Explicit includes mostly first-order interpolation elements.
    - Exceptions: second-order triangular and tetrahedral elements, second-order beam elements.

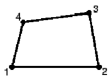
Introduction

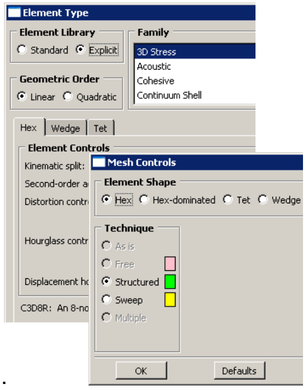
- All elements include a variety of element-based loads, for example:
  - Body loads (e.g., gravity)
  - Surface pressure loads on solid and shell elements
  - Force per unit length loads on beam elements and shell element edges
- All elements are suitable for geometrically nonlinear analysis.
  - Large displacements and rotations.
  - Large strain, except for the small-strain shell elements.
- There are no general restrictions on the use of particular material behaviors with a particular element type.
  - Any combination that makes sense is acceptable.
- Most ABAQUS/Explicit element types are also available in ABAQUS/Standard.
  - Many of these elements are discussed in detail in the *Element Selection in ABAQUS/Standard* lecture notes.

Solid Elements

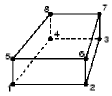
Solid Elements

- **Quadrilateral and hexahedral elements are the recommended solid (continuum) elements.**
  - Two-dimensional quadrilateral elements:
    - CPE4R (plane strain)
    - CPS4R (plane stress)
    - CAX4R (axisymmetric)





  - Three-dimensional hexahedral element:
    - C3D8R

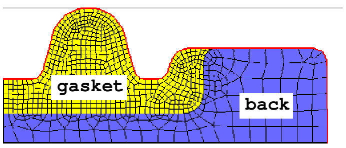


  - These are all linear, reduced-integration elements.
  - A family of corresponding coupled temperature-displacement elements is also available.

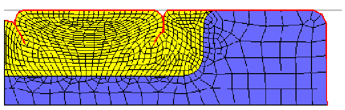
Solid Elements

- Example: Rubber gasket modeled with plane strain elements.

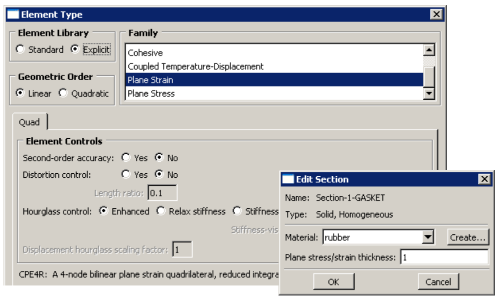
```
*ELEMENT, TYPE=CPE4R
 1, 817, 816, 815, 823
  ...
*SOLID SECTION, ELSET=gasket, MATERIAL=RUBER
 1,
*SOLID SECTION, ELSET=back, MATERIAL=PLASTIC
 1,
```



Undeformed gasket

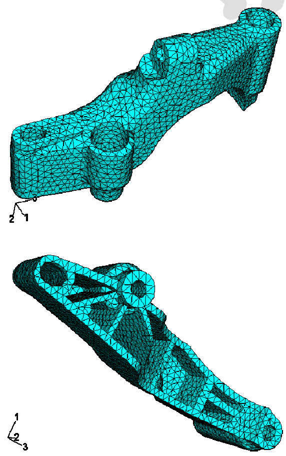


Compressed gasket



Solid Elements

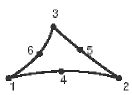
- **Tetrahedral elements**
  - Tetrahedral elements are geometrically versatile and are used in many automatic meshing algorithms.
    - However, a good mesh of hexahedral elements (C3D8R) usually provides a solution of equivalent accuracy at less cost.
  - First-order tetrahedra and triangles are usually overly stiff, and extremely fine meshes are required to obtain accurate results.
    - Avoid CPE3R, CPS3R, CAX3R, and C3D4.
    - Instead use the modified second-order tetrahedra and triangles, discussed next.

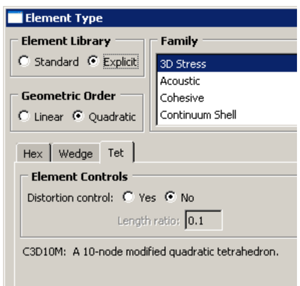


Tetrahedral mesh of a bracket

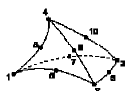
Solid Elements

- ABAQUS/Explicit uses reduced integration, first order elements **almost exclusively**.
- The exceptions include the modified 6-node triangular stress/displacement elements:
  - CPE6M
  - CPS6M
  - CAX6M





  and the modified 10-node second-order tetrahedral element
  - C3D10M.



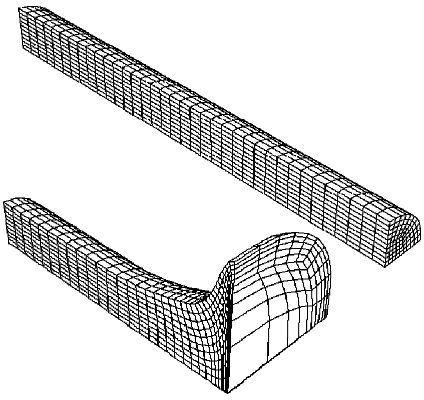
- These elements take advantage of automatic triangular and tetrahedral mesh generators and are **robust** for large deformation problems and contact.

Solid Elements

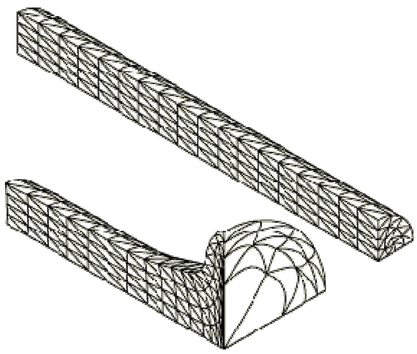
- The modified triangular and tetrahedral elements:
  - Have a **lumped mass formulation** suitable for explicit dynamic analysis
  - Suffer only **minimal** shear and volumetric locking
  - Possess a **uniform contact pressure** property that is not present in conventional second-order triangles and tetrahedra
  - Use the same formulation as ABAQUS/Standard
- When to use the modified elements:
  - Modified triangular and tetrahedral elements are effective alternatives to linear triangles and tetrahedra.
  - Quadrilateral and brick elements are preferred when such meshing is reasonable.
  - Modified elements should be used **when mesh generation dictates**.

Solid Elements

- Example: Copper rod impact into a rigid wall



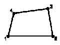
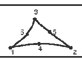
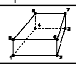
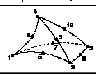
C3D8R elements



C3D10M elements

Solid Elements

– Comparison of results

Element type	Shortening (mm)	Relative CPU time	Relative Cost per Increment per Element
 **CAX4R**	**–13.11**	**1.0**	**1.0**
 **CAX6M**	**–13.13**	**1.2**	**2.91**
 **C3D8R**	**–13.10**	**11.5**	**1.86**
 **C3D10M**	**–12.71***	**22.5**	**5.83**

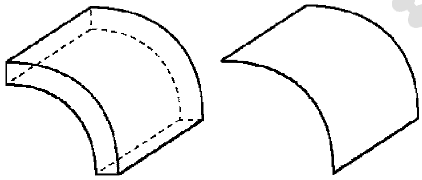
* C3D10M mesh is slightly stiffer with the given mesh refinement. The shortening value converges to –13.1 mm as the mesh is refined.

ABAQUS

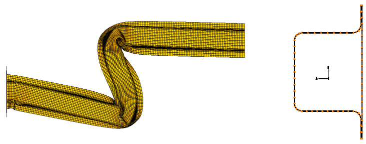
Shell and Membrane Elements

Shell and Membrane Elements

- **Shell elements**
  - Shell theory approximates a three-dimensional continuum with a two-dimensional theory.
  - This reduction in dimensionality is achieved by taking advantage of the fact that the shell is thin:
    - i.e., the thickness of the shell is small compared to typical dimensions in the shell surface.



3-D continuum surface model



Video Clip

Deformed model and cross-section for a thin-walled, energy absorbing curved beam.

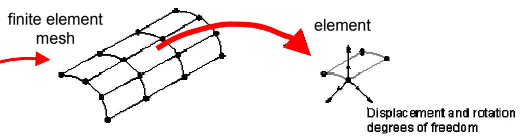
Courtesy of Honda R&D

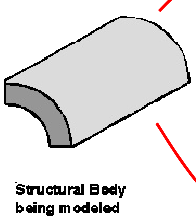
 ABAQUS

Shell and Membrane Elements

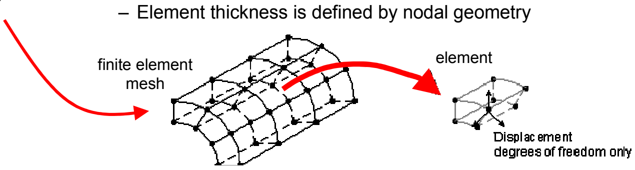
– ABAQUS/Explicit offers conventional shell elements and continuum shell elements.

- **Conventional shell model**
  - Geometry is specified at the reference surfaces.
  - Thickness is defined by section property





- **Continuum shell model**
  - Full 3-D geometry is specified
  - Element thickness is defined by nodal geometry



 ABAQUS

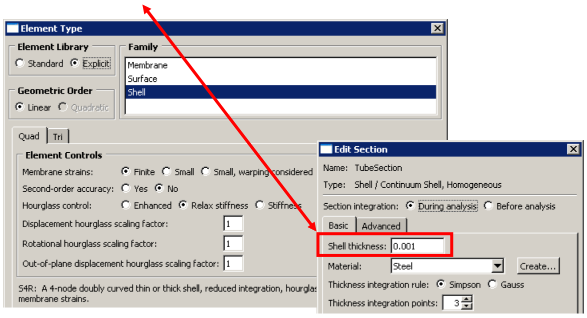
Shell and Membrane Elements

- **Conventional Shell Elements**
  - Triangular and quadrilateral conventional shell elements are available with linear interpolation and your choice of large-strain and small-strain formulations.
  - A linear axisymmetric shell element is also available.
  - For **most analyses** the standard large-strain shell elements are appropriate:
    - S4R A robust, general-purpose quadrilateral element that is suitable for a wide range of applications.
    - S3R A triangular element that may exhibit mild shear locking and it is not very accurate for curved shells.
    - SAX1 A 2-node axisymmetric shell element with three degrees of freedom per node (u_r, u_z, ϕ_θ).

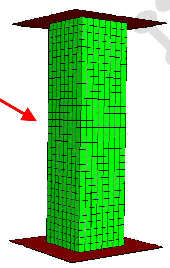
Shell and Membrane Elements

- Example: Tube crush

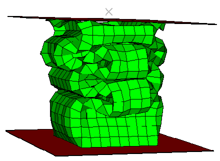
```
*ELEMENT, TYPE=S4R
 1,   1,   9, 189,  84
 2,   9,  10, 190, 189
 ...
*SHELL SECTION, ELSET=TUBE, MATERIAL=STEEL
0.001, 3
```



Element set **TUBE**



Undeformed model



Deformed model

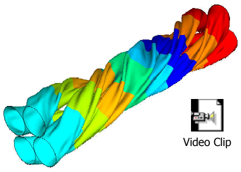
Shell and Membrane Elements

– If the analysis involves small membrane strains and arbitrarily large rotations, the small-strain shell elements are more computationally efficient:

- S4RS An efficient quadrilateral shell element; however, it can perform poorly when warped (e.g., in twisting problems)
- S4RSW More expensive than S4RS, this quadrilateral shell includes additional terms to account for warped configurations.
- S3RS A triangular shell element based on the formulation of S4RS

– Small-strain shell elements are efficient for problems that involve small membrane strains and arbitrarily large rotations.

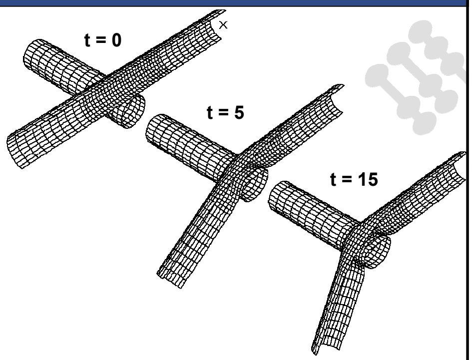
- Example: Twisting of 4 pipes:
  – There is little membrane deformation;
  – however, the pipes wrinkle and fold with high curvature.



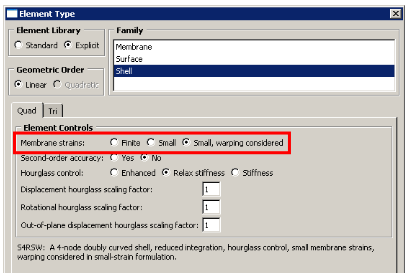
Shell and Membrane Elements

– Example: Pipe whip simulation

- Three models are analyzed, each with a different 4-node shell element type.
- Results of the three models are approximately the same;
  – however, CPU times differ.

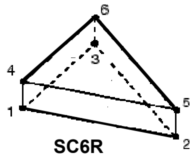


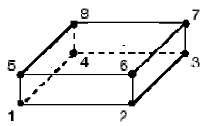
Element type	Relative CPU time
S4R	**1.0**
S4RSW	**0.78**
S4RS	**0.66**



Shell and Membrane Elements

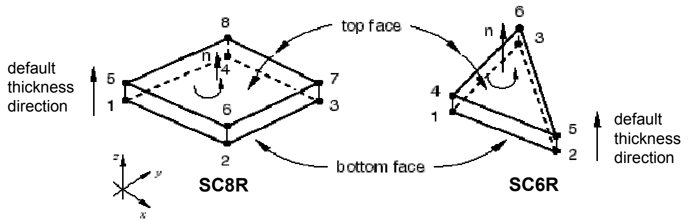
- **Continuum Shell Elements**
  - Continuum shell elements allow for:
    - Thickness tapering.
    - More accurate contact modeling than conventional shells.
      - They take into account two-sided contact and thickness changes.
    - Stacking.
      - They capture more accurately the through-thickness response for composite laminate structures.
  - Two finite-strain, general-purpose continuum shell elements are available:
    - SC6R
    - SC8R





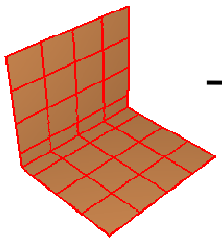
Shell and Membrane Elements

- The kinematic response in the thickness direction is different from that in the in-plane directions for the continuum shell.
- The thickness direction can be ambiguous for the SC8R element.
  - Any of the 6-faces could be the bottom face.
- By default, nodal connectivity defines the element thickness direction.
  - Nondefault directions can be specified using the STACK DIRECTION parameter on the shell section definition.

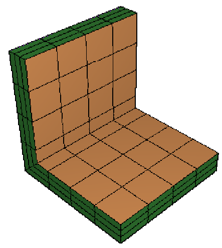


Shell and Membrane Elements

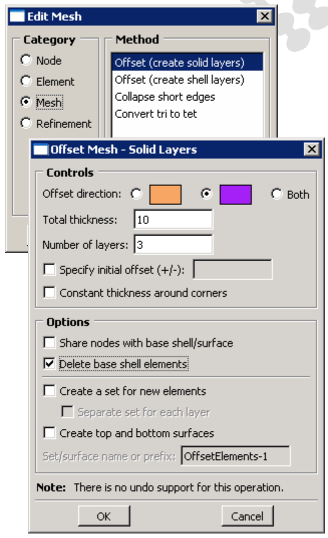
– You can create an oriented mesh in ABAQUS/CAE by offseting a shell mesh to generate layers of brick elements.



shell orphan mesh

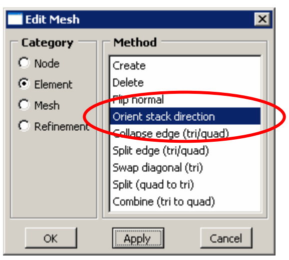


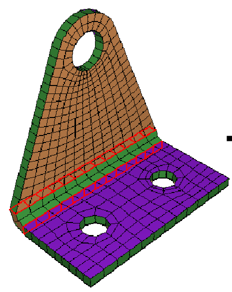
layers of brick elements



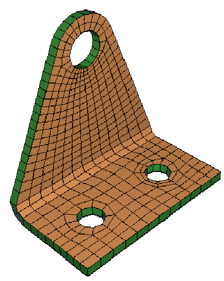
Shell and Membrane Elements

– ABAQUS/CAE also allows you to change the element stack orientation.
  - Selected elements are oriented with respect to a reference top face.
  - Node labels, element labels, and node coordinates are not altered.
  - The tool is only available for orphan meshes.
    – Orphan mesh parts can be created from meshed native geometry.





Inconsistent stack orientation

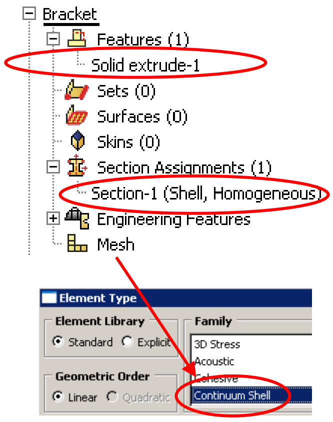


Consistent stack orientation

Shell and Membrane Elements

– The user interface looks like the interface for continuum solid elements (where appropriate) or conventional shell elements (where appropriate).

```
*element, type=SC6R, elset=triangles
*element, type=SC8R, elset=quads
*shell section, elset=triangles,
  material=steel, poisson=v,
  thickness modulus=e
*shell section, elset=quads, composite,
  orientation=orient,
  stack direction = {1|2|3|orientation}
thickness, # sect pts, material, orientation
*material, name=steel
*elastic
*plastic
  …
```

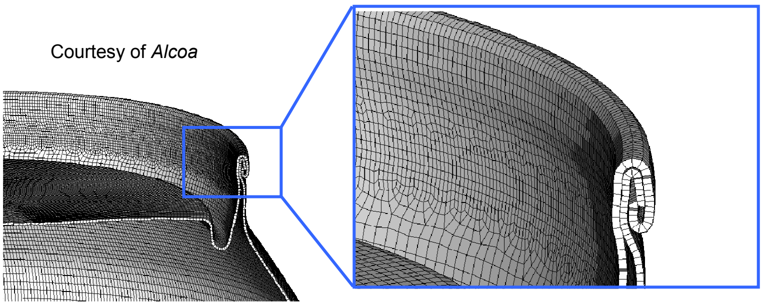


Shell and Membrane Elements

– Example: Can forming problem

- Here we are modeling the process that forms the lip/seam between the top of the can and the sidewall.
  – Difficulties are encountered using conventional shell elements.
  – The problem is readily solved with continuum shell elements.

Courtesy of *Alcoa*

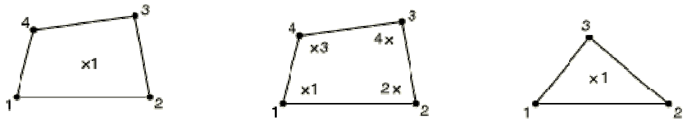


Shell and Membrane Elements

- In ABAQUS/Explicit the element stable time increment can be controlled by the continuum shell element thickness.
  - A continuum shell element model may take significantly more increments to complete than a comparable conventional shell element model.
  - The small stable time increment size may be mitigated by specifying a lower stiffness in the thickness direction when appropriate.

Shell and Membrane Elements

- **Membrane elements in ABAQUS/Explicit**
  - Membrane elements (M3D4R, M3D4, M3D3) are used to represent thin surfaces in space that offer strength in the plane of the element but have no bending stiffness.
    - For example, thin rubber sheet that forms a balloon or airbag
  - M3D4R are 4-node quadrilateral, reduced integration elements with hourglass control
  - M3D4 are 4-node quadrilateral, fully integrated elements (no hourglassing)

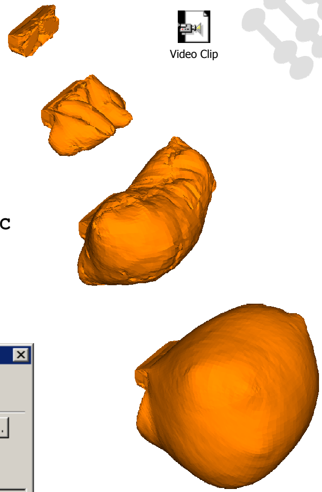


M3D4R M3D4 M3D3

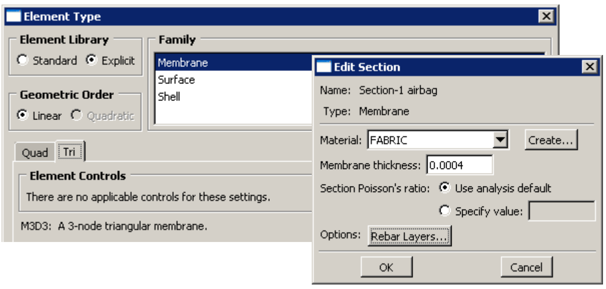
Shell and Membrane Elements

– Example: Single chamber airbag

```
*ELEMENT, TYPE=M3D3, ELSET=AIRBAG
        1,        21,        10,        20
        2,        37,        23,        36
        3,        54,        38,        53
        ...
*MEMBRANE SECTION, ELSET=AIRBAG, MATERIAL=FABRIC
 0.4E-3,
```



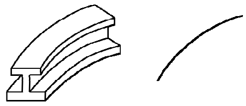
Video Clip

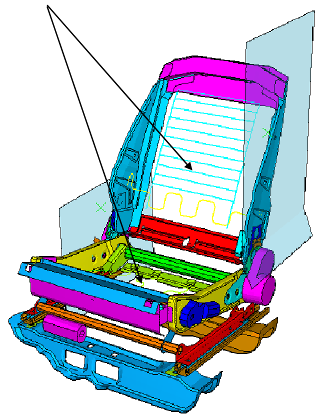


Beam and Truss Elements

Beam and Truss Elements

- **Beam elements in ABAQUS/Explicit**
  - Beam theory approximates a three-dimensional continuum with a one-dimensional theory.
  - Slenderness assumption: the dimensions in the cross-section of the beam are assumed small compared to typical dimensions along the beam axis.





*Courtesy of BMW**

* Gholami, T., J. Lescheticky, and R. Paßmann, "Crashworthiness Simulation of Automobiles with ABAQUS/Explicit," ABAQUS Users' Conference, Munich, 2003.

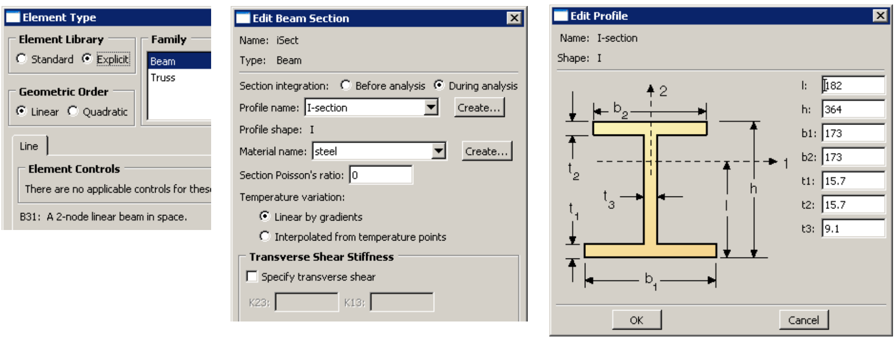
Beam and Truss Elements

- A beam element is a one-dimensional line element that has stiffness associated with deformation of the beam's "axis."
- These deformations include axial stretch, curvature change (bending), and—in three-dimensional space—torsion.
- The beam elements available in ABAQUS/Explicit offer additional flexibility associated with transverse shear deformation between the beam's axis and its cross-section.
  - These elements are called *shear flexible* or *Timoshenko* beams.
- The main advantages of beam elements are:
  - They are geometrically simple.
  - They have few degrees of freedom.
- Three-dimensional beam element types:
  - B31 - shear flexible with linear interpolation (most commonly used)
  - B32 - shear flexible with quadratic interpolation.

Beam and Truss Elements

– Example: I-beam with a standard universal section, UB 356 × 171

```
*Element, type=B31
 1,  1,  2
*BEAM SECTION, ELSET=CANT, SECTION=I, MATERIAL=STEEL
** l,  h,   b1,   b2,   t1,   t2,   t3
182., 364., 173., 173., 15.7, 15.7, 9.1
0., 0., -1.
```



Beam and Truss Elements

- **Truss elements in ABAQUS/Explicit**
  - Truss elements are rods that can carry only tensile or compressive loads.
    - They have no resistance to bending; therefore, they are useful for modeling pin-jointed frames.
    - When a beam is very slender, it can be modeled as a truss.
  - The following truss element is available in three dimensions:
    - T3D2

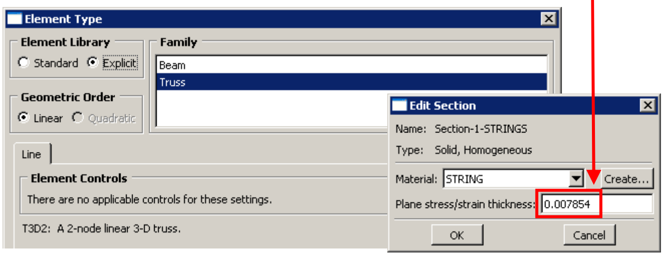
Tennis racket

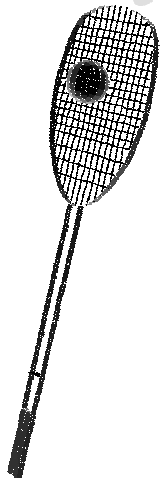
Beam and Truss Elements

– Example: Tennis racket

```
*ELEMENT,TYPE=T3D2, ELSET=STRINGS
   1, 204, 205
  11, 303, 304
  ...
*SOLID SECTION, ELSET=STRINGS, MATERIAL=STRING
7.854E-3,
```

string cross-sectional area





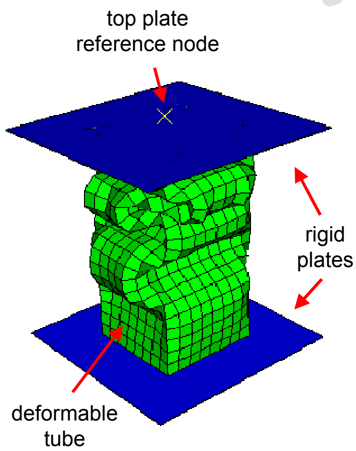
ABAQUS

ABAQUS

Rigid Bodies

Rigid Bodies

- **ABAQUS/Explicit has a general rigid body capability.**
  - A rigid body is a collection of nodes and elements whose motion is governed by the motion of a single reference node.
  - Any body or part of a body can be defined as a rigid body.
- **Rigid bodies are computationally efficient.**
  - Their motion is described completely by no more than six degrees of freedom at the reference node.
- **It may be useful to make parts of a model rigid for model verification purposes.**
  - This is discussed further Lecture 10, *Managing Large Models.*



Tube crush example

Rigid Bodies

- **A rigid body definition consists of at most:**

  1 Rigid body reference node (required)

  1 Element set
  - This set may contain deformable and/or rigid elements.

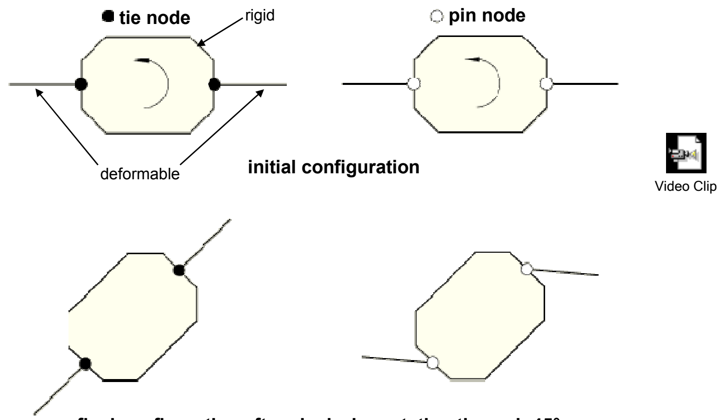
  1 Analytical rigid surface
  - Three-dimensional analytical rigid surfaces are obtained by revolving or extruding a two-dimensional geometric profile.

  1 Pin node set
  - Pin nodes have their translational degrees of freedom only associated with the rigid body.

  1 Tie node set
  - Tie nodes have both their translational and rotational degrees of freedom associated with the rigid body.

Rigid Bodies

- Example demonstrating pin vs. tie connections to rigid bodies



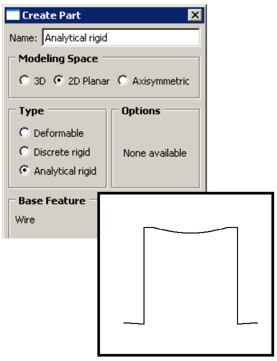
final configuration after clockwise rotation through 45°

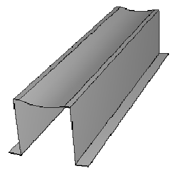
Rigid Bodies

- **Analytical rigid surfaces**
  - Analytical rigid surfaces provide a smoother surface profile than a surface discretized with rigid elements.
  - Advantages:
    - Have the potential to reduce noise in the solution significantly.
    - May be computationally efficient when compared to rigid surfaces made from element faces.
    - Can be easier to define.
  - Disadvantages:
    - Cannot be used to define a general three-dimensional rigid geometry.
      - Defined by revolving or extruding a two-dimensional profile
    - Cannot display contact pressure distribution;
      - can recover reaction force at rigid body reference node and pressure distribution on slave surface.

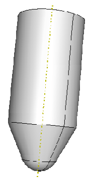
Rigid Bodies

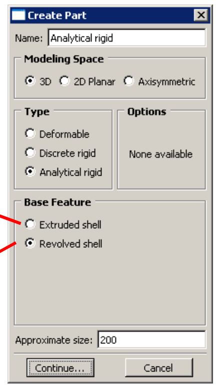
– Three types of analytical rigid surfaces are available in ABAQUS (SEGMENTS, CYLINDER, and REVOLUTION).





CYLINDER type
(an extruded 3-D surface)





SEGMENTS type
(2-D surface)

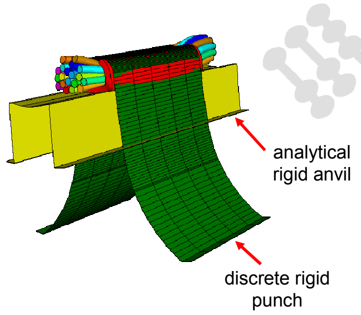
REVOLUTION type
(a revolved 3-D surface)

Rigid Bodies

– Analytical rigid surfaces cannot be used with:
  - Spot welds
  - Small sliding contact
  - General contact
    – Contact with analytical rigid surfaces must be defined through contact pair interactions.
    – Discussed further in Lecture 4, *Contact Modeling*.

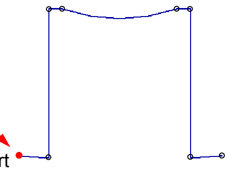
Rigid Bodies

- **Example: Wire crimping**



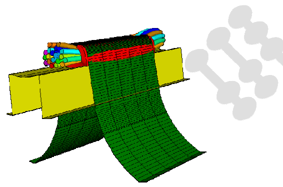

```
*ELEMENT, TYPE=R3D4,ELSET=PUNCH
  1,  1,  2, 35, 42
  2, 42, 35, 36, 41
  ...
*RIGID BODY, REF NODE=20000, ELSET=PUNCH
```

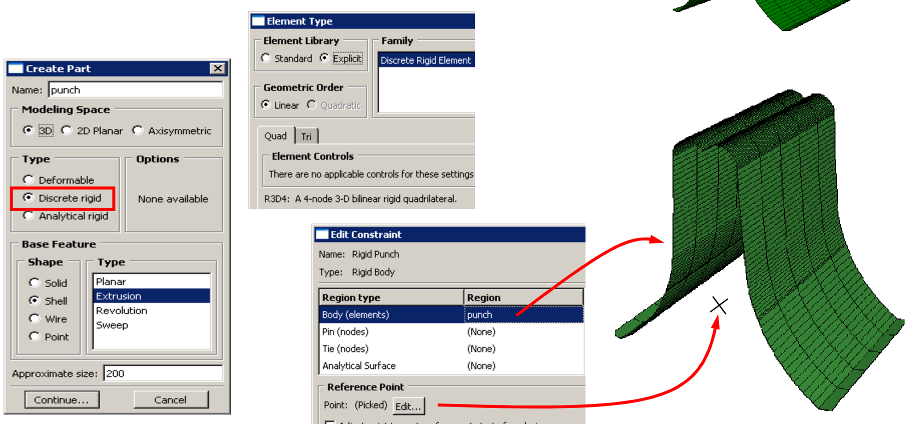
```
*SURFACE, TYPE=CYLINDER, NAME=ANVIL
  START, -1.50, -1.05
   LINE, -1.04, -1.08
   LINE, -1.04, 1.08
   LINE, -0.83, 1.08
   CIRCL, 0.83, 1.08, 0., 3.60
   LINE, 1.04, 1.08
   LINE, 1.04, -1.08
   LINE, 1.50, -1.05
*RIGID BODY, REF NODE=10000, ANALYTICAL SURFACE=ANVIL,
 POSITION=CENTER OF MASS
```



Rigid Bodies

- **Example (cont'd): Wire crimping**

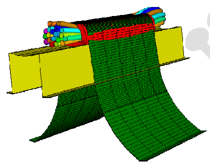


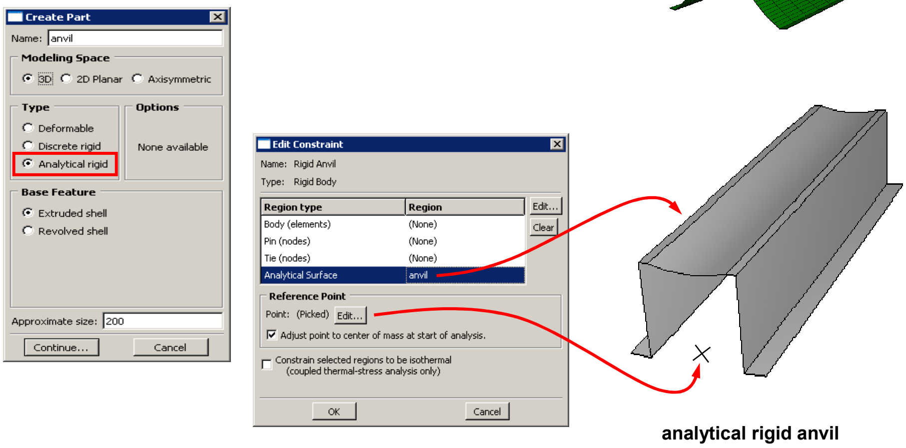


discrete rigid punch

Rigid Bodies

- **Example (cont'd): Wire crimping**



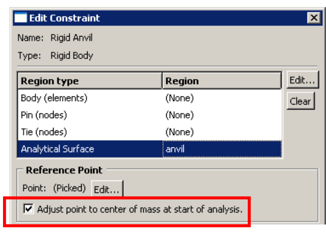


analytical rigid anvil

Rigid Bodies

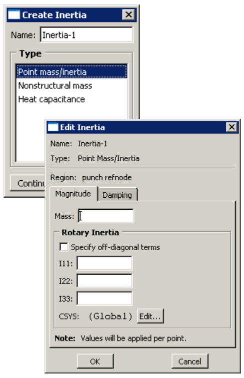
- **Location of the rigid body reference node**
  - You may place the rigid body reference node anywhere in a model.
  - The location is important if the rigid body is to move freely under applied loads during the analysis.
    - In this case, the node should be placed at the center of mass.
  - ABAQUS/Explicit can calculate the center of mass and relocate the reference node to this location automatically.
    - In this case, the new coordinates of the reference node are also printed out at the end of the printed output file.
  - Syntax:

```
*RIGID BODY, REF NODE=<node>,
 ELSET=<element set>, POSITION=CENTER OF MASS
```



Rigid Bodies

- **Inertial properties of rigid bodies**
  - Inertial properties must be defined for any rigid body that can move freely under applied loads.
    - i.e., rigid bodies that are not fully constrained
  - ABAQUS calculates the inertial properties of a discrete rigid body based on the contributions from its elements.
  - MASS and ROTARYI elements can be defined at the rigid body reference node and slave nodes.
  - The inertial properties of each rigid body are printed out at the end of the printed output (.dat) file, including:
    - mass,
    - center of mass, and
    - moments of inertia about the center of mass.



Rigid Bodies

- It is possible to define the thickness and density of rigid elements on the ∗RIGID BODY option.

```
*RIGID BODY, REF NODE=<node>, ELSET=<element set>, DENSITY=#
<thickness>
```

  - A constant thickness can be specified as a value on the data line following the ∗RIGID BODY option, as indicated above.
  - A variable thickness can be specified by using the NODAL THICKNESS parameter on the ∗RIGID BODY option.

ABAQUS

Special-Purpose Elements

Special-Purpose Elements

- **ABAQUS/Explicit has a number of special-purpose elements, including:**
  - Mass and rotary inertia elements
    - Used to specify inertial properties at discrete points
  - Nonstructural mass
    - Used to model features that contribute to the model mass but have negligible structural stiffness.
  - Surface elements
    - Versatile elements that model surfaces in space.
  - Connector elements and cohesive elements
    - Used to model connections between regions of a model.
    - These elements are discussed in Lecture 7, *Constraints and Connections*.

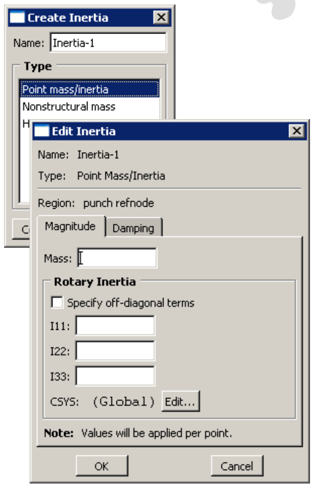
Special-Purpose Elements

- **Mass and rotary inertia elements**
  - These elements define mass and rotary inertia at a discrete point.
  - Element types:
    - MASS point mass
    - ROTARYI rotary inertia at a point
  - Use the *MASS option to define the element property for a mass element:

```
*MASS, ELSET=<element set name>
<mass magnitude>,
```

  - Use the *ROTARY INERTIA option to define the element properties for a rotary inertia element:

```
*ROTARY INERTIA, ELSET=<element set name>
<I11>, <I22>, <I33>,<I12>, <I13>, <I23>
```



 ABAQUS

Special-Purpose Elements

- **Nonstructural mass**
  - Certain physical features are often omitted from due to their negligible structural stiffness. Examples include:
    - Paint
    - Fuel in a tank
  - However, their mass contribution may be significant and should often be included in the model for accuracy.
  - Nonstructural mass can be negative.
    - For example, mass can be removed to account for bolt holes in an approximate model.
  - With nonstructural mass, the user can:
    - add or remove mass,
    - either locally over a certain region of the model or uniformly over the entire model.

 ABAQUS

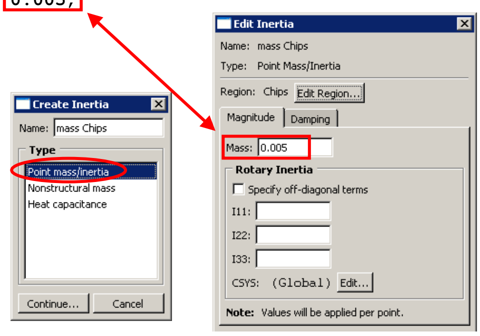
Special-Purpose Elements

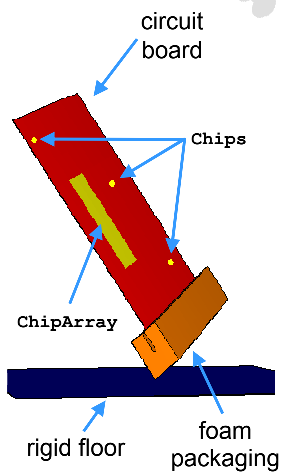
- **Different ways of adding nonstructural mass**
  - Nonstructural mass is smeared over a model region (ELSET).
    - This element set can contain solid, shell, membrane, surface, beam, or truss elements.
  - The nonstructural mass can be specified in the following forms:
    - a total mass value,
    - a mass per unit volume,
    - a mass per unit area (for elements sets that contain conventional shell, membrane, and/or surface elements), or
    - a mass per unit length (for element sets that contain beam and/or truss elements).
  - When a “total mass” is specified, it can be distributed among elements either in proportion to their volume or structural mass.
  - A mixture of valid element types can be used with each specification.

Special-Purpose Elements

- **Example: Circuit board drop test**

```
*ELEMENT, TYPE=MASS, ELSET=Chips
 6001, 60
 6002, 357
 6003, 403
*MASS,ELSET=Chips
0.005,
```

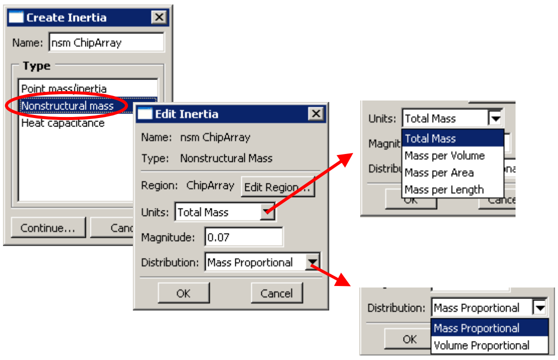


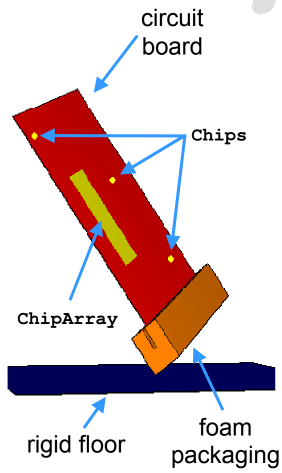


Special-Purpose Elements

- **Example: Circuit board drop test**

```
*NONSTRUCTURAL MASS, ELSET=ChipArray,
 UNITS=TOTAL MASS, DISTRIBUTION=MASS PROPORTIONAL
0.07,
```

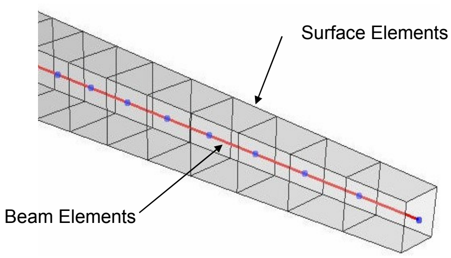




Special-Purpose Elements

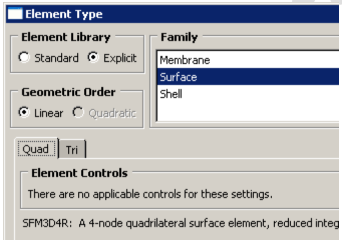
- **Surface elements**
  - Surface elements have no inherent stiffness and behave just like membrane elements with zero thickness.
    - Surface elements can transmit only in-plane forces.
  - They can be used to specify a complex surface on beam elements when used in conjunction with a surface-based tie constraint.

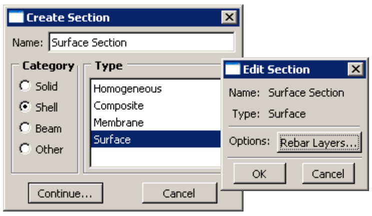
Surface elements tied **(*TIE)** to beam elements to model exterior contact surface



Special-Purpose Elements

- There are two surface element types.
  - Triangular surface element:
    - SFM3D3
  - Quadrilateral surface element with reduced integration:
    - SFM3D4R
- Cross-sectional properties of surface elements are defined with the *SURFACE SECTION option.





ABAQUS

ABAQUS

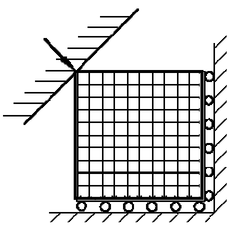
Hourglassing

Hourglassing

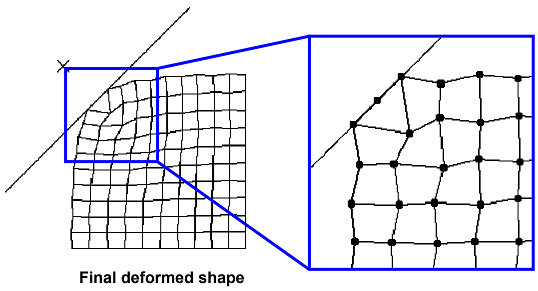
- **In most general problems the stiffness and mass of an element must be calculated numerically.**
  - The numerical algorithm used to integrate these variables influences how an element behaves.
- **ABAQUS/Explicit primarily uses reduced integration for first-order elements; that is, only one integration point per element.**
  - These elements are cheap and effective.
  - They minimizes the computational expense of element calculations.

Hourglassing

- **The use of the reduced-integration scheme has a drawback: it can result in mesh instability, commonly referred to as "hourglassing."**
  - The hourglass mode does not cause any strain and, hence, does not contribute to the energy integral.
    - It behaves in a manner that is similar to that of a rigid body mode.
  - Example: Rubber block compressed diagonally by a rigid surface.



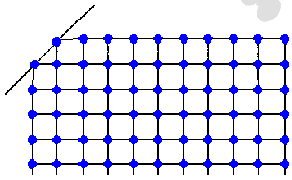
Rubber block model



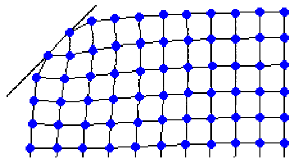
Final deformed shape

Hourglassing

- **Four common causes of excessive hourglassing and their remedies**
  - Concentrated force at a single node
    - Remedy: distribute the force among several nodes or apply a distributed load.
  - Boundary condition at a single node
    - Remedy: distribute the boundary constraint among several nodes.
  - Contact at a single node
    - Remedy: distribute the contact constraint among several nodes.
  - Bending with too few elements
    - Remedy: use at least 4 elements through the section of bending regions.



undeformed

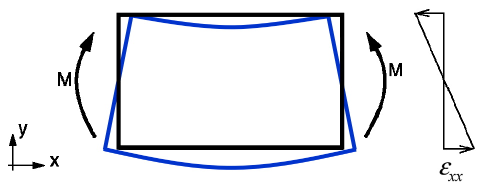


deformed

Rounded corner prevents hourglassing of rubber block.

Hourglassing

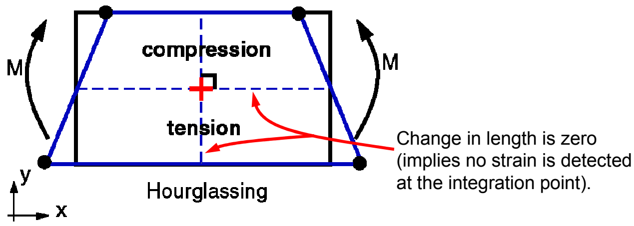
- **Consider the physical characteristics of pure bending.**
  - The assumed behavior of the material that finite elements attempt to model is:
    - Plane cross-sections remain plane throughout the deformation.
    - The axial strain ε_{xx} varies linearly through the thickness.
    - The strain in the thickness direction ε_{yy} is zero if $\nu = 0$.
    - No membrane shear strain.
      - Implies that lines parallel to the beam axis lie on a circular arc.



Hourglassing

- **Reduced-integration low-order elements (e.g., CPS4R) have only one integration point.**
- **These elements have the following bending behavior:**
  - The single element should detect strain, but it does not.
  - The deformation is a spurious zero-energy mode.
    - Deformation but no strain—hourglassing
      - also called "keystoning" because of the trapezoidal shape.

Bending behavior for a single first-order reduced-integration element.

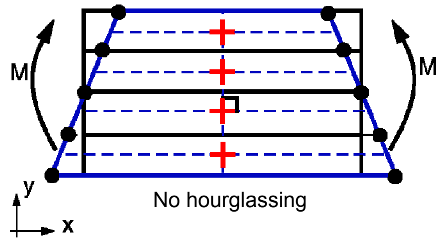


 ABAQUS

Hourglassing

- **Hourglassing is not a problem if you use multiple elements—at least four through the thickness.**
  - Each element captures either compressive or tensile axial strains, but not both.
  - The axial strains are measured correctly.
  - The thickness and shear strains are zero.
  - Cheap and effective elements.

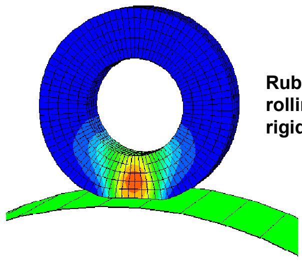
Bending behavior with four elements through the thickness



 ABAQUS

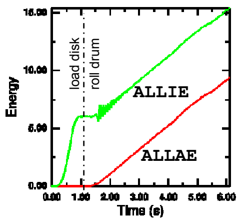
Hourglassing

- **When is hourglassing a problem?**
  - Hourglassing is **almost never** a problem with the enhanced hourglass control available in ABAQUS.
    - More robust than other schemes
    - No user-set parameters
    - Based on enhanced strain methods

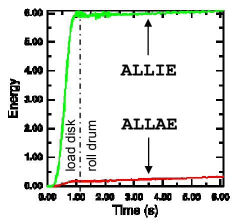


Rubber disk rolling against rigid drum

Combined hourglass control scheme



Enhanced hourglass control scheme



Comparison of energy histories

Hourglassing

- To activate enhanced hourglass control, use the option

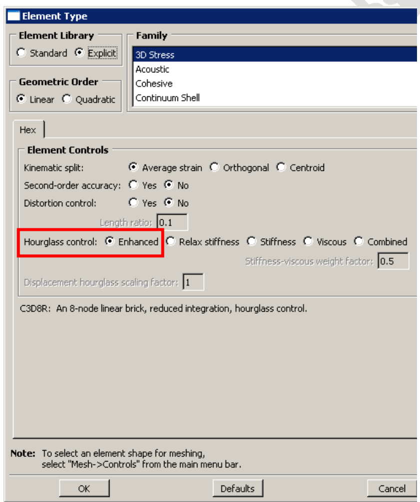
  ***SOLID SECTION, CONTROLS=name, ELSET=elset**

  ***SECTION CONTROLS, NAME=name, HOURGLASS=ENHANCED**

  No user parameters

- ABAQUS/CAE usage:

  Mesh module: Mesh→Element Type



Hourglassing

- **Currently, enhanced hourglass control is not the default scheme for most elements.**

ABAQUS/Explicit hourglass control methods
Relax stiffness (default for most elements)
Enhanced strain
Stiffness
Viscous
Combined (stiffness+viscous)

– Enhanced hourglass control is the default for:
  - All modified triangle and tetrahedral elements
  - All elements modeled with finite-strain elastic materials (hyperelastic and hyperfoam)

Hourglassing

- **Excessive use of hourglass control energy can be identified by looking at the energy histories.**

– Verify that the artificial energy used to control hourglassing is small ($<1\%$) relative to the internal energy.
  - Use the energy output quantities ALLAE and ALLIE to verify this.

– The exception to this rule is the case of elastic bending with a coarse mesh when enhanced hourglass control is used.
  - This is discussed next.

Hourglassing

- **Elastic bending problems and coarse mesh accuracy**
  - For elastic bending problems, improved coarse mesh accuracy may be obtained using the enhanced hourglass control method.
    - The enhanced hourglass control formulation is tuned to give highly accurate results for elastic bending in the presence of coarse meshes.
    - In this particular case (elastic bending+coarse mesh+enhanced hourglass control), the hourglass energy may be higher than the recommended limit.
      - This does not necessarily mean the results are adversely affected; however, you must use engineering judgment to assess the validity of the results.

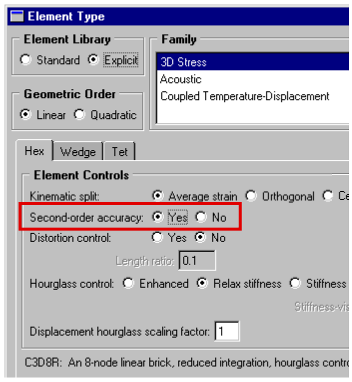
ABAQUS/Explicit: Advanced Topics

ABAQUS

Second-order Accuracy

Second-order Accuracy

- **Accuracy order of the formulation**
  - ABAQUS/Explicit offers both first-and second-order accurate formulation options for solid and shell elements.
  - First-order accuracy is the default.
    - Sufficient accuracy for nearly all problems.
  - Second-order accuracy is usually required for analyses with components undergoing a large number of revolutions (>5).
  - Usage:




```
*SECTION CONTROLS, NAME=name, SECOND ORDER ACCURACY=YES
```


Second-order Accuracy

- **Example: spinning propeller**