Element Selection Criteria

Element Selection Criteria

Overview

- Elements in ABAQUS
- Structural Elements (Shells and Beams) vs. Continuum Elements
- Modeling Bending Using Continuum Elements
- Stress Concentrations
- Contact
- Incompressible Materials
- Mesh Generation
- Solid Element Selection Summary

Elements in ABAQUS

• The wide range of elements in the ABAQUS element library provides flexibility in modeling different geometries and structures.

– Each element can be characterized by considering the following:

• Family
• Number of nodes
• Degrees of freedom
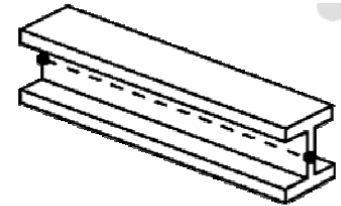• Formulation
• Integration

Elements 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.
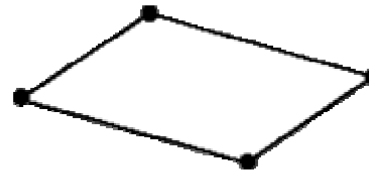
– There are many variations within a family.
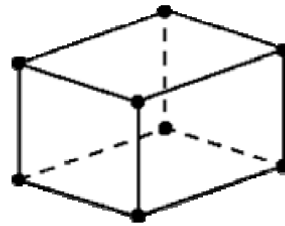
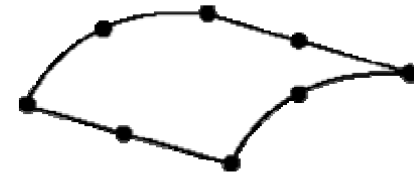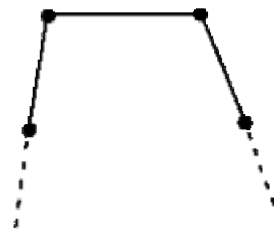continuum (solid elements)

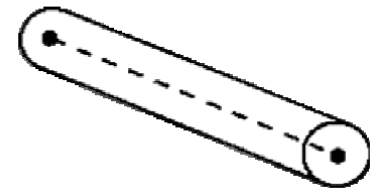shell elements

beam elements

rigid elements

membrane elements

infinite elements

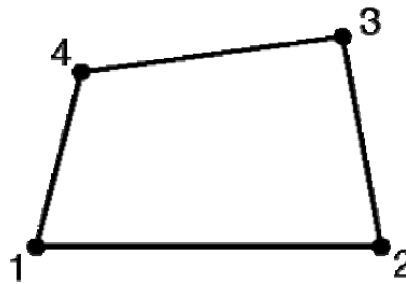special-purpose elements like springs, dashpots, and masses
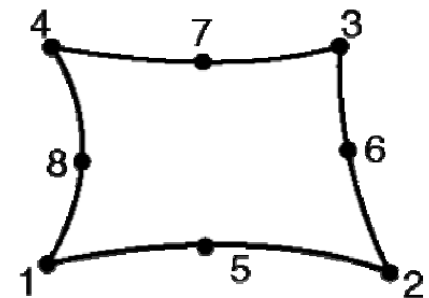
truss elements

Elements 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

Elements in ABAQUS

• Degrees of freedom

– The primary variables that exist at the nodes of an element are the degrees of freedom in the finite element analysis.

– Examples of degrees of freedom are:

• Displacements

• Rotations

• Temperature

• Electrical potential

– Some elements have internal degrees of freedom that are not associated with the user-defined nodes.

Elements in ABAQUS

• Formulation

– The mathematical formulation used to describe the
behavior of an element is another broad category that is used
to classify elements.

– Examples of different element formulations:

Plane strain   Small-strain shells

Plane stress   Finite-strain shells

Hybrid elements   Thick shells

Incompatible-mode elements   Thin shells

Elements in ABAQUS

- Integration
  - The stiffness and mass of an element are calculated umerically at sampling points called "integration points" within the element.
  - The numerical algorithm used to integrate these variables influences how an element behaves.
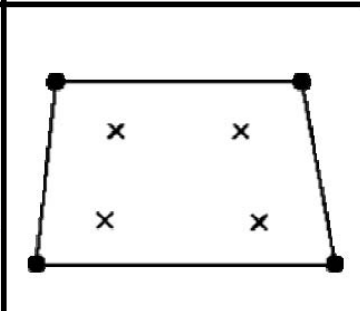
Elements in ABAQUS
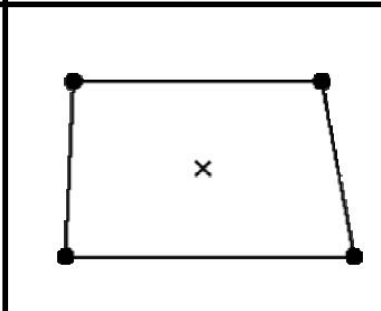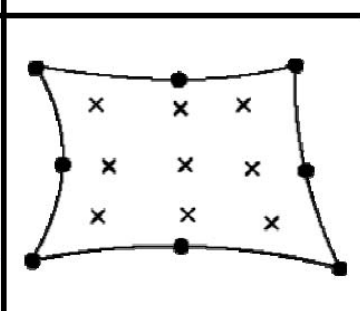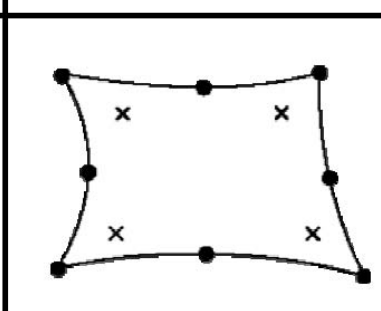
– ABAQUS includes elements with both "full" and "reduced" integration.

• **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.

	Full integration	Reduced integration
First-order interpolation		
Second-order interpolation		

Elements in ABAQUS

- Element naming conventions: examples

B21: **B**eam, **2**-D, **1**st-order interpolation

S8RT: **S**hell, **8**-node, **R**educed integration, **T**emperature

CAX8R: **C**ontinuum, **AX**isymmetric, **8**-node, **R**educed integration
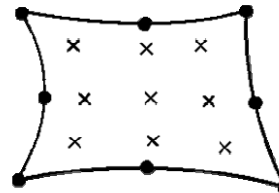
CPE8PH: **C**ontinuum, **P**lane strain, **8**-node, **P**ore pressure, **H**ybrid
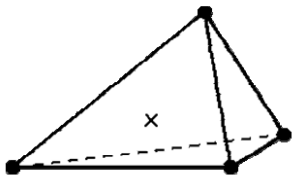
DC3D4: **D**iffusion (heat transfer), **C**ontinuum, **3**-D, **4**-node

DC1D2E: **D**iffusion (heat transfer), **C**ontinuum, **1**-D, **2**-node, **E**lectrical

Elements in ABAQUS

• Comparing ABAQUS/Standard and ABAQUS/Explicit element libraries

– Both programs have essentially the same element families: continuum, shell, beam, etc.

– ABAQUS/Standard includes elements for many analysis types besides stress analysis: heat transfer, soils consolidation, acoustics, etc.

    • Acoustic elements are also available in ABAQUS/Explicit.

– ABAQUS/Standard includes many more variations within each element family.

– ABAQUS/Explicit includes mostly first-order integration elements.

    • Exceptions: second-order triangular and tetrahedral elements and second-order beam elements.

–Many of the same general element selection guidelines apply to both programs.

Structural Elements (Shells and Beams) vs. Continuum Elements

• Continuum (solid) element models can be large and expensive, particularly in three-dimensional problems.

• If appropriate, structural elements (shells and beams) should be used for a more economical solution.

> – A structural element model typically requires far fewer elements than a comparable continuum element model.

• For structural elements to produce acceptable results, the shell thickness or the beam cross-section dimensions should be less than 1/10 of a typical global structural dimension, such as:

> – The distance between supports or point loads
> – The distance between gross changes in cross section
> – The wavelength of the highest vibration mode

Structural Elements (Shells and Beams) vs. Continuum Elements

- Shell elements

  - Shell elements approximate a three-dimensional continuum with a surface model.
    - Model bending and in-plane deformations efficiently.
  - If a detailed analysis of a region is needed, a local three dimensional continuum model can be included using multi-point constraints or submodeling.
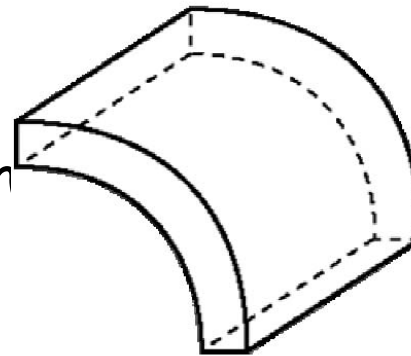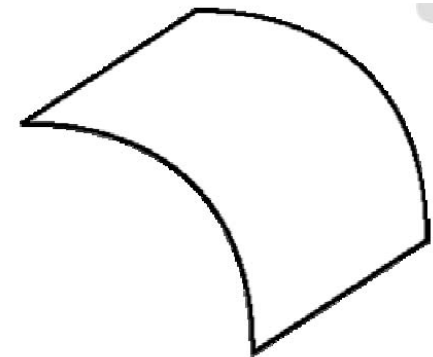
3-D continuum   surface model
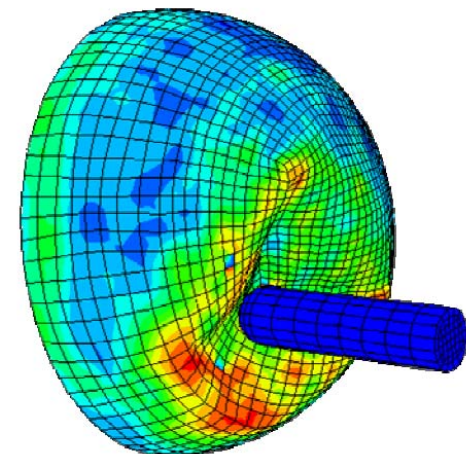
shell model of a hemispherical dome subjected to a projectile impact

Structural Elements (Shells and Beams) vs. Continuum Elements

- **Beam elements**

  - Beam elements approximate a three-dimensional continuum with a line model.
    - Model bending, torsion, and axial forces efficiently.
    - Many different cross-section shapes are available.
    - Cross-section properties can also be specified by providing engineering constants.
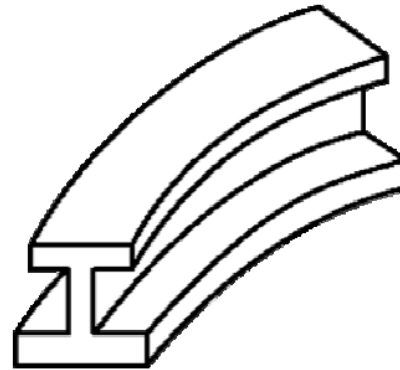
3-D continuum   line model
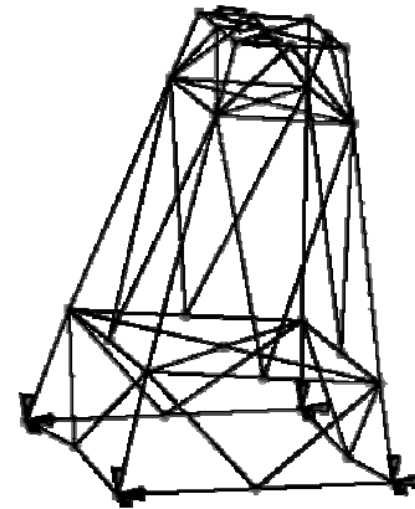
framed structure modeled
using beam elements

Modeling Bending Using Continuum Elements

• Physical characteristics of pure bending

– This is the assumed behavior of the
material that finite elements attempt
to model.

• 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 $v = 0$.
• No membrane shear strain.
– Implies that lines parallel to
the beam axis lie on a circular
arc.
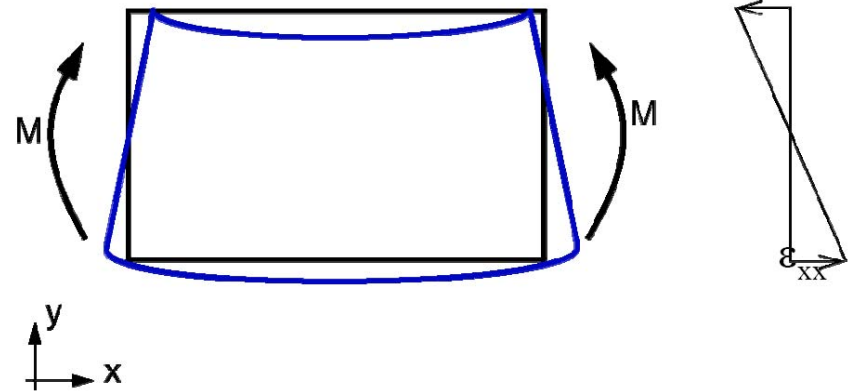
Modeling Bending Using Continuum Elements

- Modeling bending using second-order solid elements (CPE8, C3D20R, …)

  – Second-order full- and reduced-integration solid elements model bending accurately:

    – The axial strain equals the change in length of the initially horizontal lines.

    – The thickness strain is zero.

    – The shear strain is zero.

lines that are initially vertical do not change length (implies $\varepsilon_{yy} = 0$).

M   M

Isoparametric lines

y

x
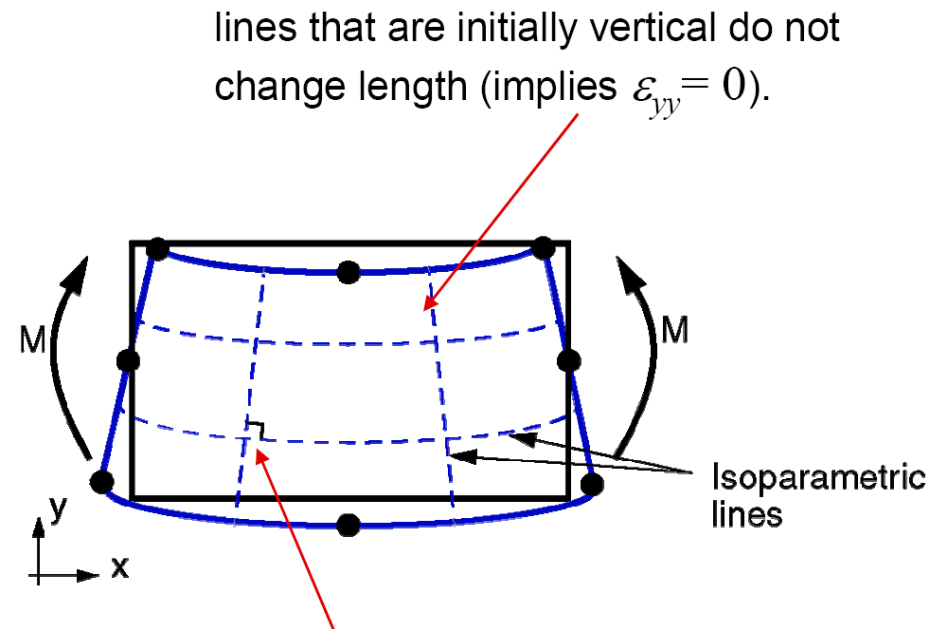
Because the element edges can assume a curved shape, the angle between the deformed isoparametric lines remains equal to 90° (implies $\varepsilon_{xy} = 0$).

Modeling Bending Using Continuum Elements

• Modeling bending using first-order fully integrated solid elements (CPS4, CPE4, C3D8)

- These elements detect shear strains at the integration points.
  • Nonphysical; present solely because of the element formulation used.
- Overly stiff behavior results from energy going into shearing the element rather than bending it (called "shear locking").

integration point

M   M

≠90°

y

x

Shear Locking

Because the element edges must remain straight, the angle between the deformed isoparametric lines is not equal to 90° (implies $\varepsilon_{xy} \neq 0$).
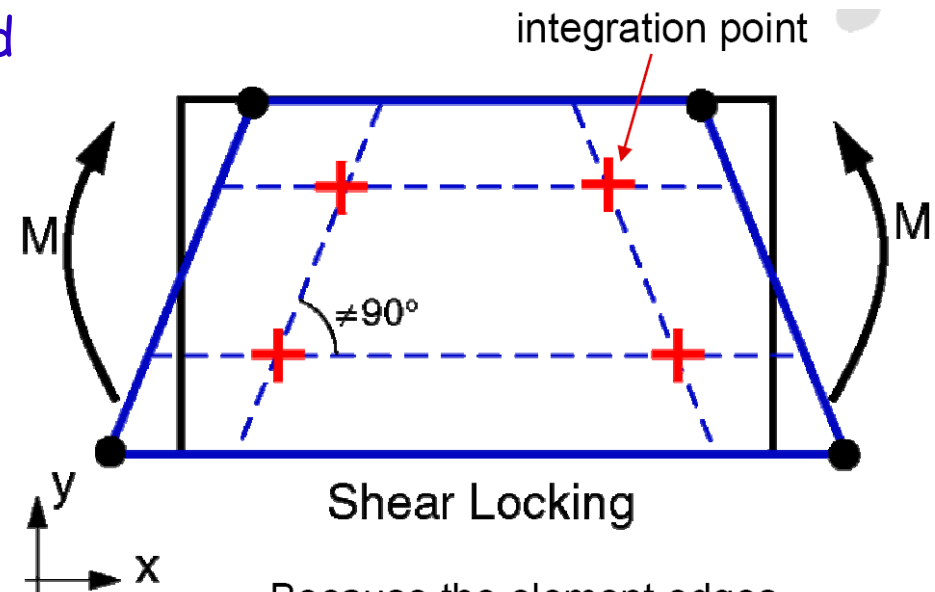
Do not use these elements in regions dominated by bending!

Modeling Bending Using Continuum Elements

• Modeling bending using first-order reduced-integration elements (CPE4R, …)

– These elements eliminate shear locking.

– However, hourglassing is a concern when using these elements.

  • Only one integration point at the centroid.

  • A single element through the thickness does not detect strain in bending.

  • Deformation is a zero-energy mode (deformation but no strain; called "hourglassing").

Change in length is zero (implies no strain is detected at the integration point).

compression

M   M

tension

y

x

Hourglassing

Bending behavior for a single first-order reduced-integration element
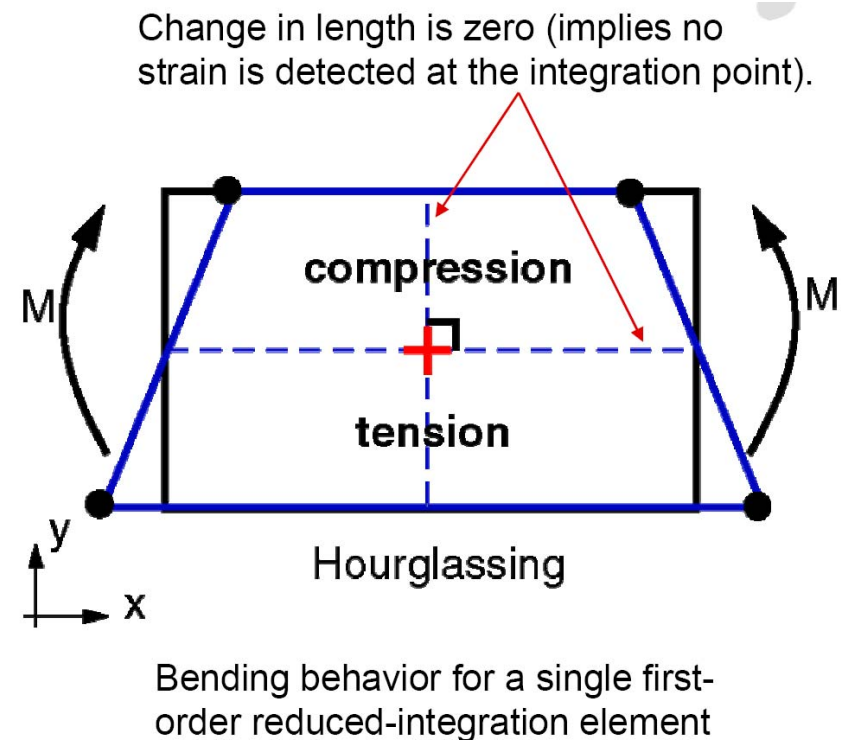
Modeling Bending Using Continuum Elements

– Hourglassing can propagate easily
  through a mesh of first-order reduced-
  integration elements, causing unreliable
   results.

–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.

Four Elements through the Thickness
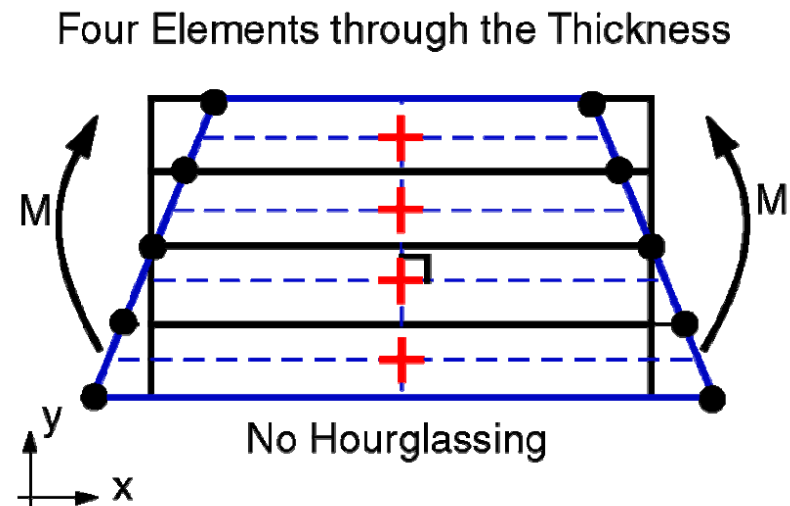
No Hourglassing

Modeling Bending Using Continuum Elements

• Detecting and controlling hourglassing

– Hourglassing can usually be
    seen in   deformed shape
    plots.
    • Example: Coarse and
      medium meshes of a
      simply supported beam
      with a center point load.
  – ABAQUS has built-in
    hourglass controls that limit
    the problems caused by
    hourglassing.
    • Verify that the
      artificial energy used to
      control hourglassing is
      small (<1%) relative to
      the internal energy.
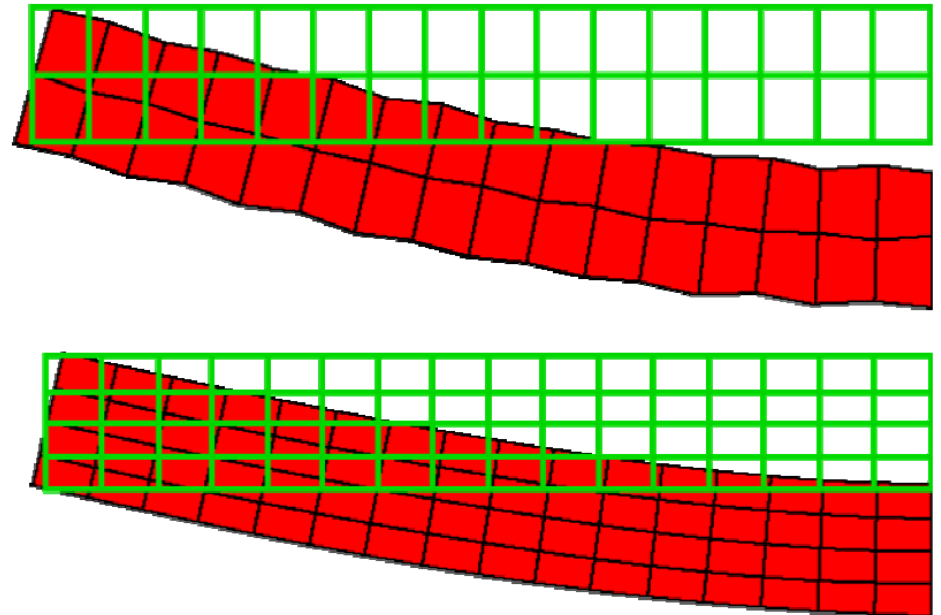
Same load and displacement
magnification (1000×)

Modeling Bending Using Continuum Elements

– Use the X–Y plotting capability in ABAQUS/Viewer to compare the energies graphically.
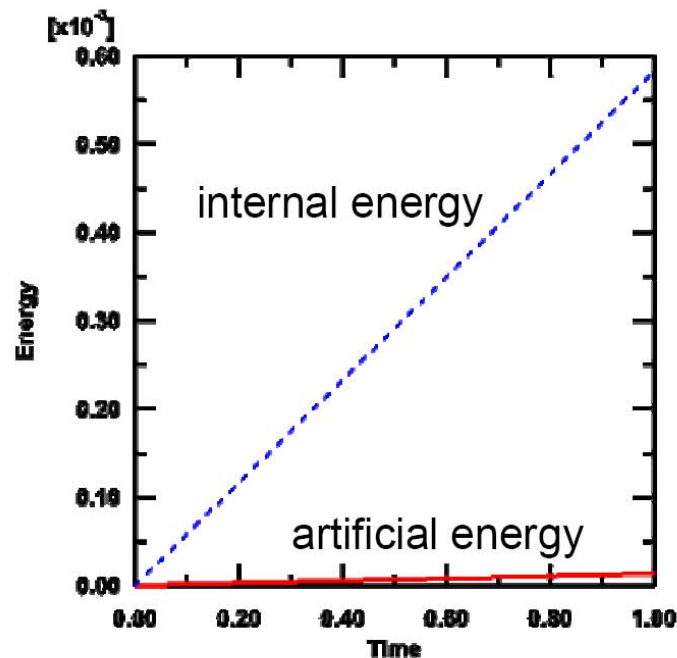
Two elements through the thickness: Ratio of artificial to internal energy is 2%

Four elements through the thickness: Ratio of artificial to internal energy is 0.1%

Modeling Bending Using Continuum Elements

- Modeling bending using incompatible mode elements (CPS4I, …)
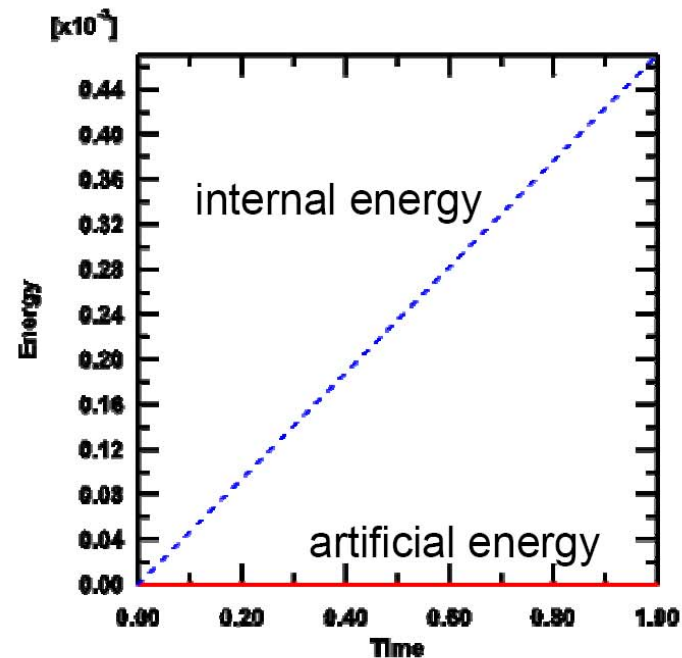- Perhaps the most cost-effective solid continuum elements for bending dominated problems.
- Compromise in cost between the first- and second-order reduced integration elements, with many of the advantages of both.
  - Model shear behavior correctly—no shear strains in pure bending.
  - Model bending with only one element through the thickness.
  - No hourglass modes, and work well in plasticity and contact problems.
- The advantages over reduced-integration first-order elements are reduced if the elements are severely distorted; however, all elements perform less accurately if severely distorted.

Modeling Bending Using Continuum Elements

- Example: Cantilever beam with distorted elements
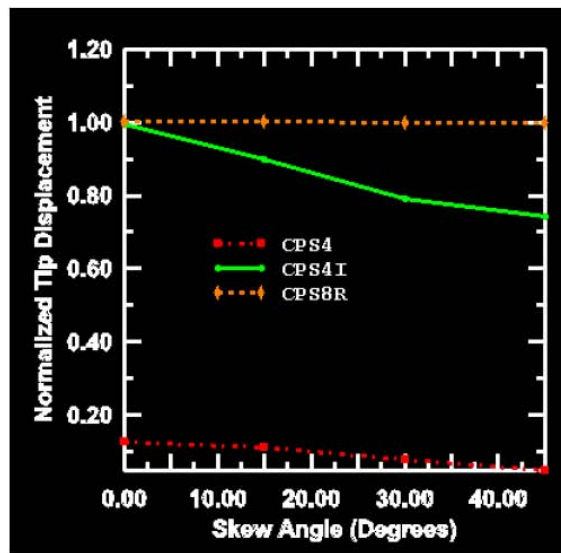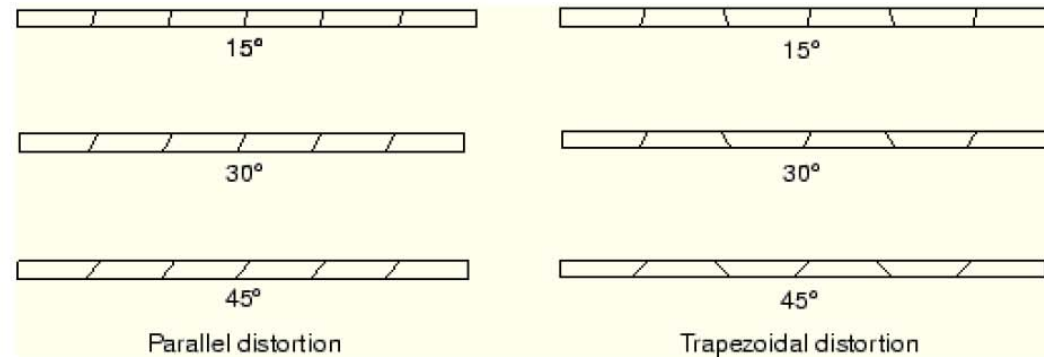
Parallel distortion
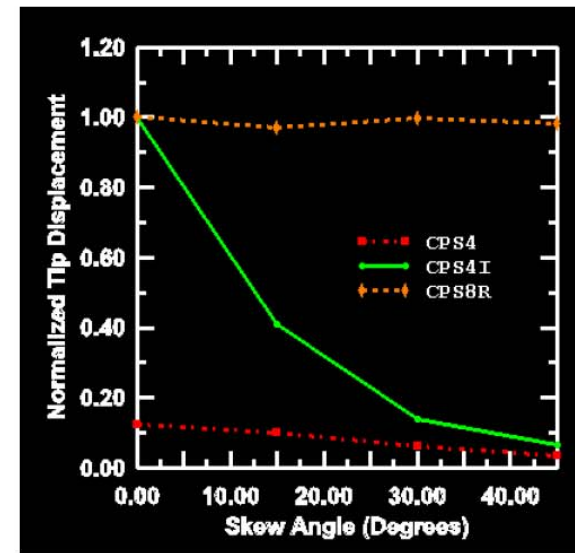
Trapezoidal distortion

Modeling Bending Using Continuum Elements

- Summary

Element type	ε_{xx}	ε_{yy}	ε_{xy}	Notes
Physical behavior	$\neq 0$	0	0	
Second-order	$\neq 0$	0	0	OK
First-order, full integration	$\neq 0$	$\neq 0$	$\neq 0$	Shear locking
First-order, reduced integration	0	0	0	Hourglassing if too few elements through thickness
	$\neq 0$	0	0	OK if enough elements through the thickness
Incompatible mode	$\neq 0$	0	0	OK if not overly distorted

Stress Concentrations

• Second-order elements clearly outperform first-order elements in problems with stress concentrations and are ideally suited for the analysis of (stationary) cracks.

– Both fully integrated and reduced-integration elements work well.

– Reduced-integration elements tend to be somewhat more efficient—results are often as good or better than full integration at lower computational cost.

Stress Concentrations

– Second-order elements capture geometric features, such as curved edges, with fewer elements than first-order elements.
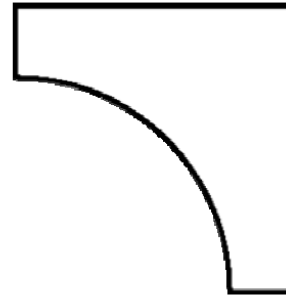
Physical model
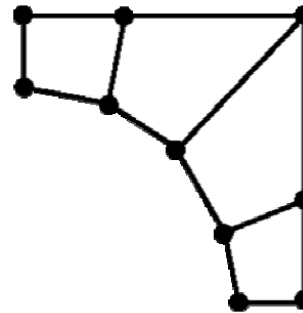
Model with first-order elements—element faces are straight line segments
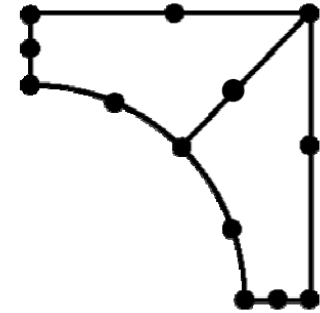
Model with second-order elements— element faces are quadratic curves

Stress Concentrations

– Both first- and second-order quads
and bricks become less accurate when
their initial shape is distorted.

- First-order elements are known
  to be less sensitive to distortion
  than secondorder elements and,
  thus, are a better choice in
  problems where significant mesh
  distortion is expected.

– Second-order triangles and
tetrahedra are less sensitive to initial
element shape than most other
elements; however, wellshaped
elements provide better results.

ideal   okay   bad

90°   ≠90°

undistorted   distorted
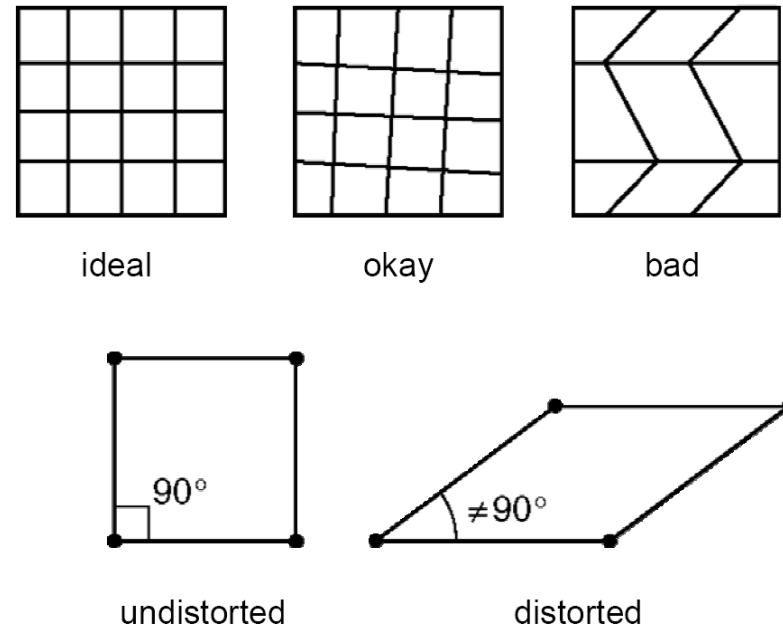
Stress Concentrations

– A typical stress concentration problem, a NAFEMS benchmark problem, is shown at right. The analysis results obtained with different element types follow.

$$\left(\frac{x}{3.25}\right)^2 + \left(\frac{y}{2.75}\right)^2 = 1$$

elliptical shape

$$\left(\frac{x}{2}\right)^2 + y^2 = 1$$

Thickness = 0.1

P

B

1.75

A

1.0

y

x

D   C

2.0   1.25
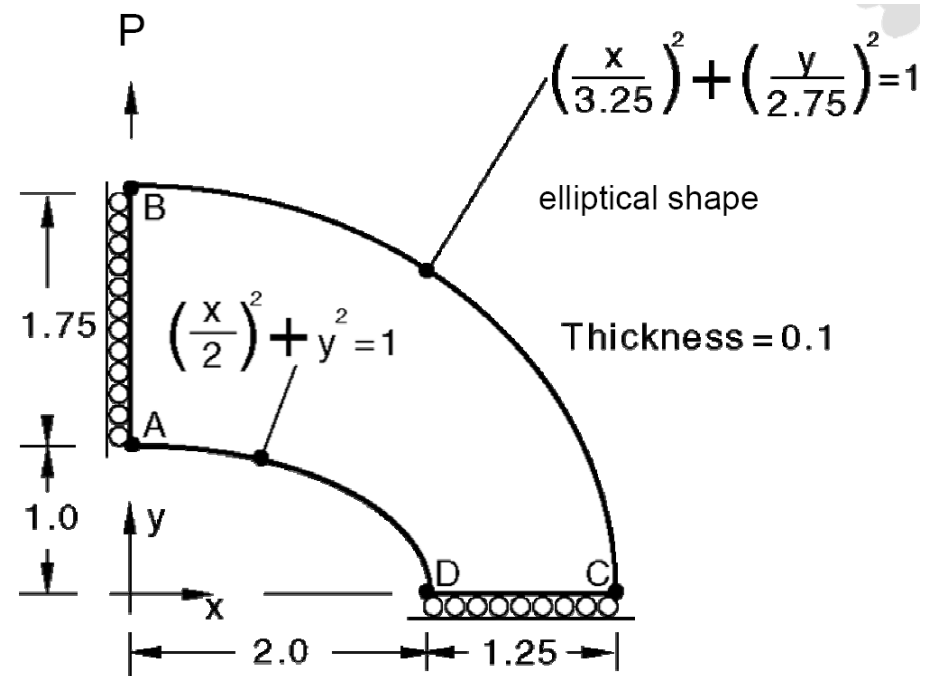
Stress Concentrations

• 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 Concentrations

– Second-order elements such as CPS6, CPS8, and CPS8R 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.
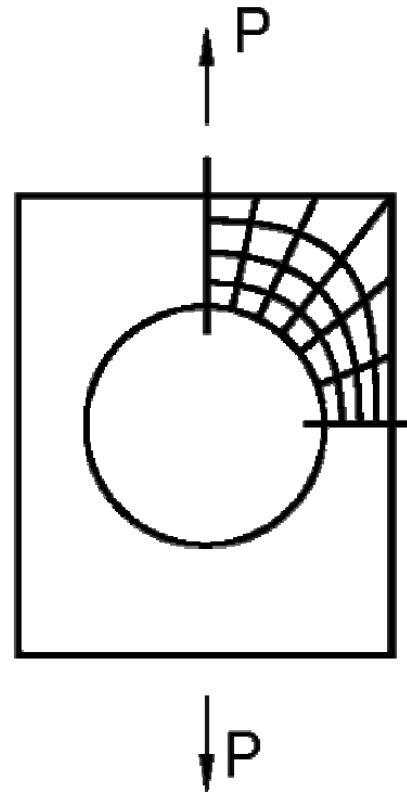
Distortion in low
strain gradient area
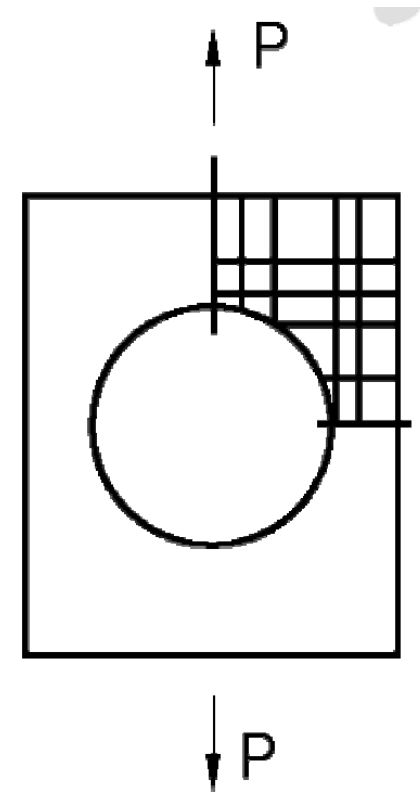
Distortion in high
strain gradient area

Contact

- Almost all element types are formulated to work well in contact problems, with the following exceptions:

  – Second-order quad/hex elements

  – "Regular" second-order tri/tet (as opposed to "modified" tri/tet elements whose names end with the letter "M"), second-order wedge, and 6- node shell and membrane elements.
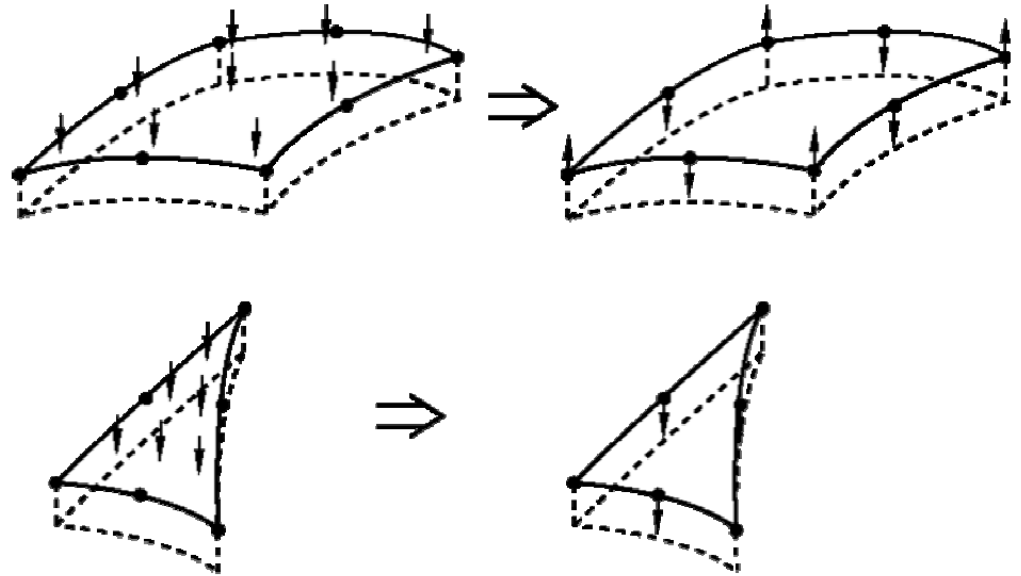
For the second-order tet, wedge, and 6-node shell and membrane elements, the directions of the consistent nodal forces resulting from a pressure load are not uniform with "classical" hard contact; use penalty-based contact instead.

- Convergence difficulties may arise with these elements.

Incompressible Materials

• Many nonlinear problems involve incompressible materials (υ = 0.5) and nearly incompressible materials (υ > 0.475).

   • Rubber
   • Metals at large plastic strains

overly stiff behavior of an elastic-plastic material with volumetric locking

correct behavior of an elastic-plastic material

Example of the effect of volumetric locking

– Conventional finite element meshes often exhibit overly stiff behavior due to volumetric locking, which is most severe when these materials are highly confined.

Incompressible Materials

– For an incompressible material each integration point's volume must remain almost constant. This overconstrains the kinematically admissible displacement field and causes volumetric locking

- For example, in a refined three-dimensional mesh of 8-node hexahedra, there is—on average—1 node with 3 degrees of freedom per element.
- The volume at each integration point must remain fixed.
- Fully integrated hexahedra use 8 integration points per element; thus, in this example, we have as many as 8 constraints per element, but only 3 degrees of freedom are available to satisfy these constraints.
- The mesh is overconstrained—it "locks."

– Volumetric locking is most pronounced in fully integrated elements.

– Reduced-integration elements have fewer volumetric constraints.

- Reduced integration effectively eliminates volumetric locking in many problems with nearly incompressible material.

Incompressible Materials

– Fully incompressible materials modeled with solid elements must use the "hybrid" formulation (elements whose names end with the letter "H").

- In this formulation the pressure stress is treated as an independently interpolated basic solution variable, coupled to the displacement solution through the constitutive theory.
- Hybrid elements introduce more variables into the problem to alleviate the volumetric locking problem. The extra variables also make them more expensive.
- The ABAQUS element library includes hybrid versions of all continuum elements (except plane stress elements, where this is not needed).

Incompressible Materials

– Hybrid elements are only necessary for:
  • All meshes with strictly incompressible materials, such as rubber.
  • Refined meshes of reduced-integration elements that still show volumetric locking problems. Such problems are possible with elasticplastic materials strained far into the plastic range.

– Even with hybrid elements a mesh of first-order triangles and tetrahedra is overconstrained when modeling fully incompressible materials. Hence, these elements are recommended only for use as "fillers" in quadrilateral or brick-type meshes with such material.

Mesh Generation

- Elements are generated in the Mesh module of ABAQUS/CAE.
- Meshes containing the element shapes shown at right can be generated.
- Most elements in ABAQUS are topologically equivalent to these shapes.
  - For example, CPE4 (stress), DC2D4 (heat transfer), and AC2D4 (acoustics) are topologically equivalent to a linear quadrilateral.

One-Dimensional

Lines

Two-Dimensional

Triangles

Quadrilaterals

Three-Dimensional

Tetrahedra

Triangular prisms (wedges)

Hexahedra
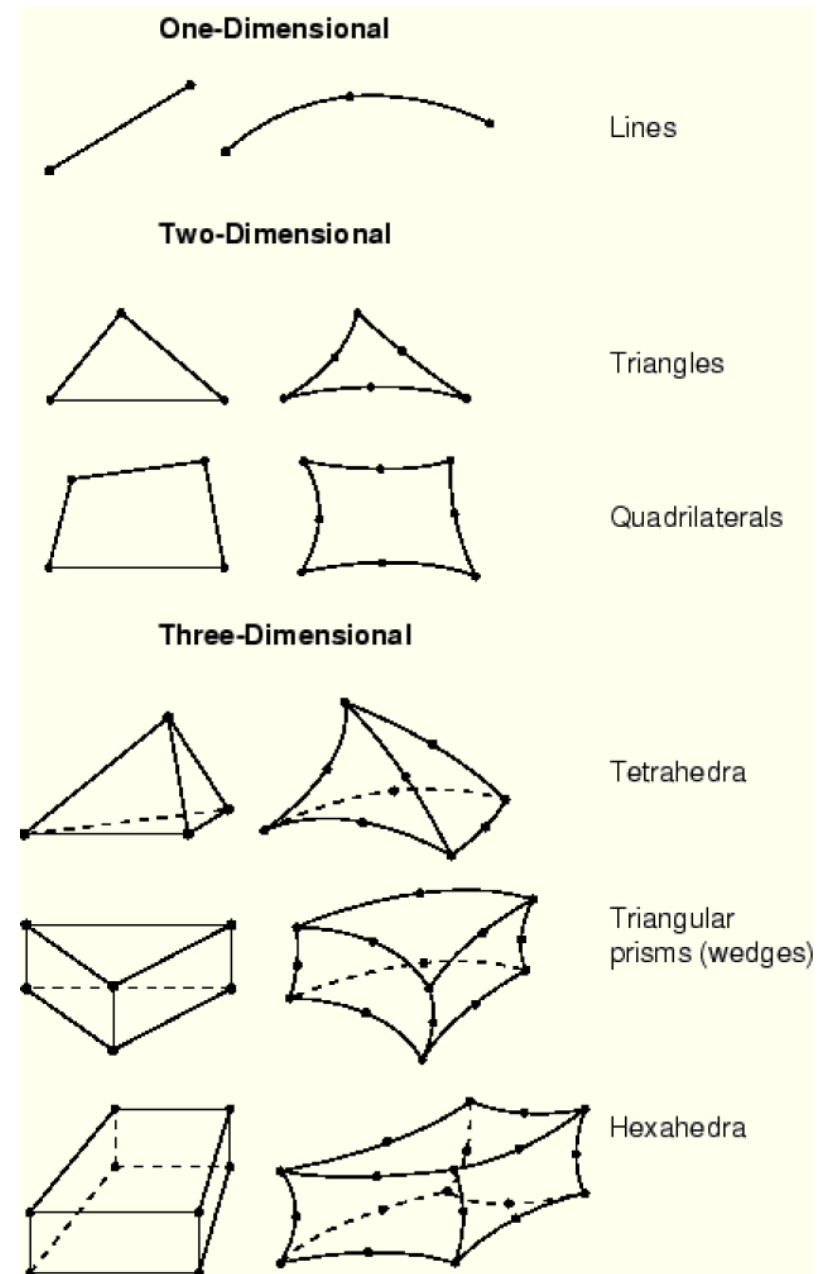
Mesh Generation

- Quad/hex vs. tri/tet elements

– Of particular importance when generating a mesh is the decision regarding whether to use quad/hex or tri/tet elements.

– Quad/hex elements should be used wherever possible.

- They give the best results for the minimum cost.
- When modeling complex geometries, however, the analyst often has little choice but to mesh with triangular and tetrahedral elements.
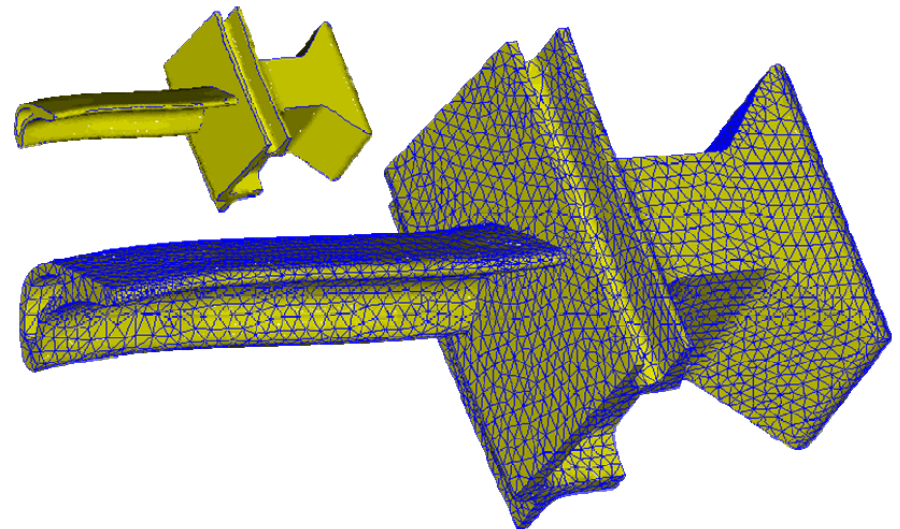
Turbine blade with platform modeled with tetrahedral elements

Mesh Generation

- First-order tri/tet elements (CPE3, CPS3, CAX3, C3D4, C3D6) are poor elements; they have the following problems:
  - Poor convergence rate.
    - They typically require very fine meshes to produce good results.
  - Volumetric locking with incompressible or nearly incompressible materials, even using the "hybrid" formulation.
- These elements should be used only as fillers in regions far from any areas where accurate results are needed.

Mesh Generation

– "Regular" second-order tet, second-
  order wedge, and 6-node shell and
  membrane elements (C3D10, C3D15,
  STRI65, M3D6) should not be used
  to model contact unless a penalty-
  based contact formulation is used.
  - Under uniform pressure the
    contact forces are significantly
    different at the corner and
    midside nodes with "classical"
    hard contact.

– Second-order triangles (CAX6, CPE6,
  CPS6) may show a noisy contact
  distribution and may cause
  convergence difficulties.
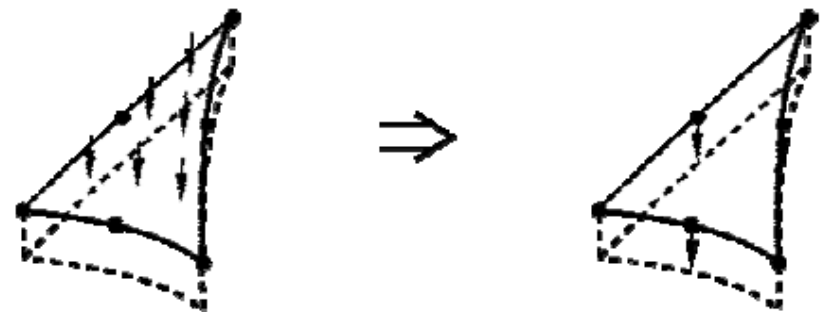
Equivalent nodal forces created by
uniform pressure on the face of a
regular second-order tetrahedral
element (C3D10) with "classical" hard
contact

Mesh Generation

- – Modified second-order tri/tet elements (C3D10M, etc.) alleviate the problems of other tri/tet elements.
  - • Good convergence rate—close to convergence rate of second-order quad/hex elements.
  - • Minimal shear or volumetric locking.
    - – Can be used to model incompressible or nearly incompressible materials in the hybrid formulation (C3D10MH).
  - • These elements are robust during finite deformation.
  - • Uniform contact pressure allows these elements to model contact accurately.

Mesh Generation

• Mesh refinement and convergence

- Use a sufficiently refined mesh to ensure that the results from your ABAQUS simulation are adequate.
    - Coarse meshes tend to yield inaccurate results.
    - The computer resources required to run your job increase with the level of mesh refinement.
- It is rarely necessary to use a uniformly refined mesh throughout the structure being analyzed.
    - Use a fine mesh only in areas of high gradients and a coarser mesh in areas of low gradients.
- Can often predict regions of high gradients before generating the mesh.
    - Use hand calculations, experience, etc.
    - Alternatively, you can use coarse mesh results to identify high gradient regions.

Mesh Generation

Some recommendations:

- Minimize mesh distortion as much as possible.
- A minimum of four quadratic elements per 90° should be used around a circular hole.
- A minimum of four elements should be used through the thickness of a structure if first-order, reduced integration solid elements are used to model bending.
- Other guidelines can be developed based on experience with a given class of problem.

Mesh Generation

- It is good practice to perform a mesh convergence study.
    - Simulate the problem using progressively finer meshes, and compare the results.
        - The mesh density can be changed very easily using ABAQUS/CAE since the definition of the analysis model is based on the geometry of the structure.
    - This will be discussed further in the next lecture.
    - When two meshes yield nearly identical results, the results are said to have "converged."
        - This provides increased confidence in your results.

Solid Element Selection Summary

Class of problem	Best element	choice Avoid using
General contact between deformable bodies	First-order quad/hex	Second-order quad/hex
Contact with bending	Incompatible mode	First-order fully integrated quad/hex or second-order quad/hex
Bending (no contact)	Second-order quad/hex	First-order fully integrated quad/hex
Stress concentration	Second-order	First-order
Nearly incompressible ($v>0.475$ or large strain plasticity $e_{pl}>10\%$)	First-order elements or second-order reduced integration elements	Second-order fully integrated

Solid Element Selection Summary

Class of problem	Best element	choice Avoid using
Completely incompressible (rubber v = 0.5)	Hybrid quad/hex, first-order if large deformations are anticipated	
Bulk metal forming (high mesh distortion)	First-order reduced integration quad/hex	Second-order quad/hex
Complicated model geometry (linear material, no contact)	Second-order quad/hex if possible (if not overly distorted) or second-order tet/tri (because of meshing difficulties)	
Complicated model geometry (nonlinear problem or contact)	First-order quad/hex if possible (if not overly distorted) or modified second-order tet/tri (because of meshing difficulties)	
Natural frequency (linear dynamics)	Second-order	Second-order fully integrated
Nonlinear dynamic (impact)	First-order	Second-order