Lesson 7: Analysis of Stiffened Composite Panels

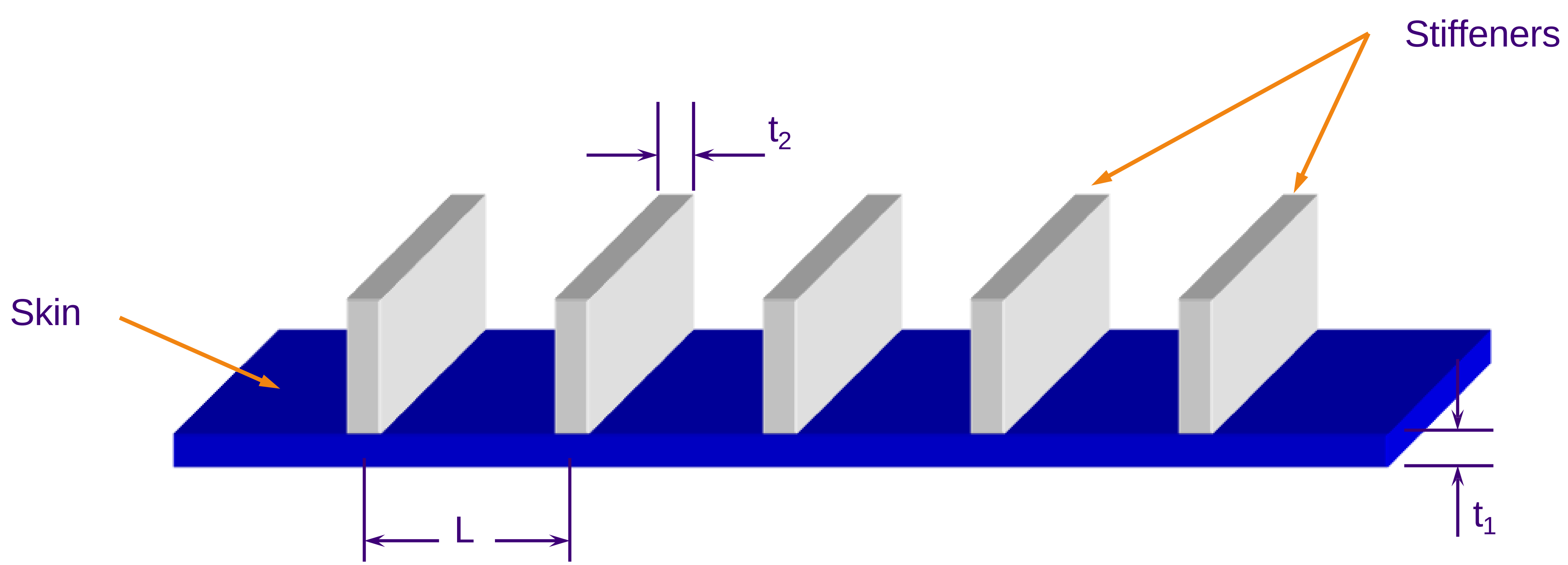
Lesson content:

- Stiffened Composite Panels
- Abaqus Usage
- Abaqus Example
- Workshop 5: Bending of a Reinforced Flat Panel under Uniform Pressure

2 hours

Stiffened Composite Panels (1/3)

- Discretely stiffened composite laminates will be the focus of this lecture.
  - This structure is created by attaching bracing elements to a composite plate or shell.
  - This structural configuration is utilized extensively in many industries (e.g., stiffening the fuselage of an aircraft).

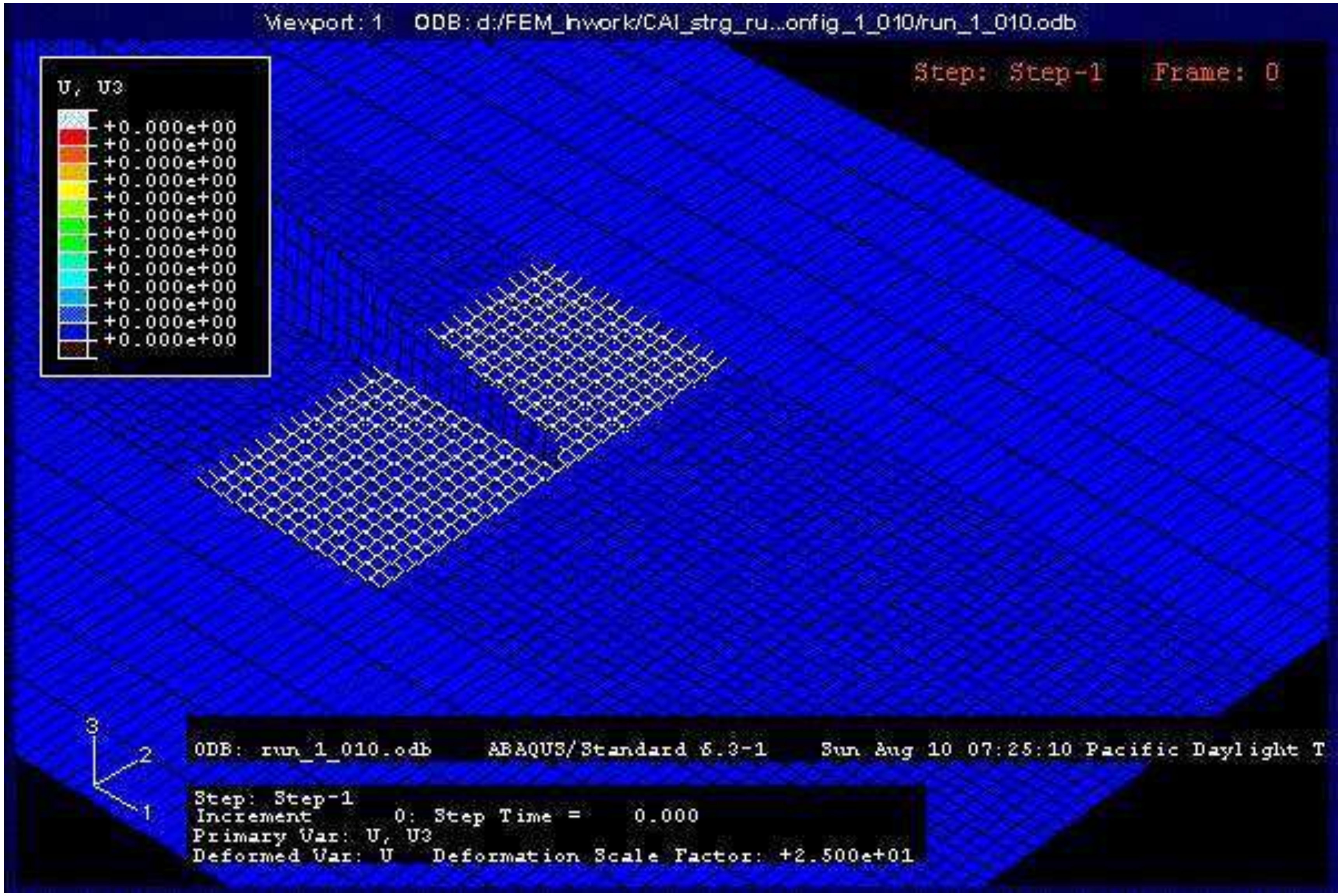


Stiffened Composite Panels (2/3)

- Many stiffened composite plates are manufactured by co-curing or adhesively bonding the stiffener to the panel.
- The connection between the skin and stiffener is usually of importance due to integrity concerns.
- Abaqus provides certain features for analyzing the failure of the interface between a skin and stringer (specifically, these techniques model progressive fracture along a path).
    - Element- and surface-based cohesive behavior
        - Discussed in Lecture 9.
    - Virtual Crack Closure Technique (VCCT)
        - Discussed in Lecture 10.
- The main purpose of this lecture, however, is to study how to actually model the skin/stiffener combination itself.
    - This will be the focus of the rest of the lecture.

Stiffened Composite Panels (3/3)

- Example of a stringer pop-off analyzed with VCCT:



Abaqus Usage (1/6)

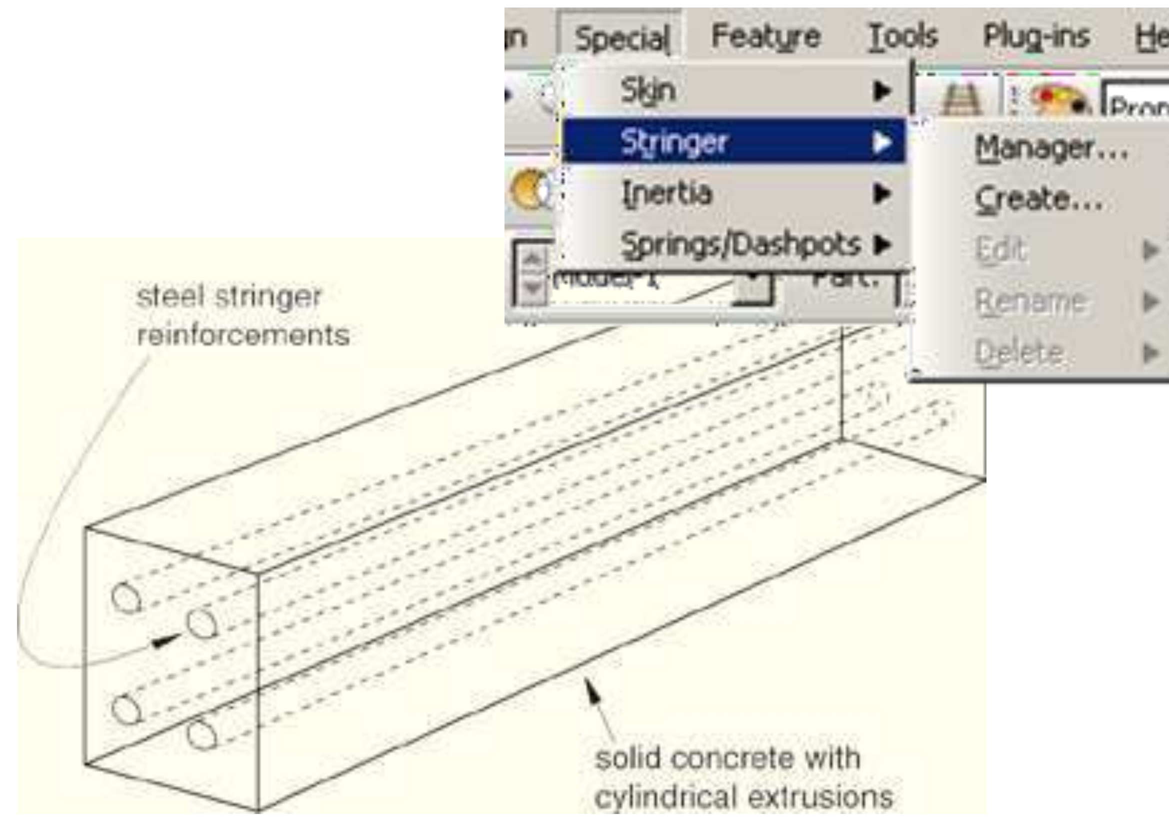
- The modeling of the skin is fairly straightforward in terms of the techniques that would be utilized.
  - The skin is typically a thin composite layup; therefore, it would be modeled utilizing a conventional or continuum shell element, or perhaps a composite solid utilizing the mixed modeling techniques described previously in Lectures 3 and 4.
- The modeling of the stiffener is the primary focus of this lecture.
- The geometry of the stiffener, as well as the pattern of the stiffener deployment, can influence the modeling strategy.
  - For example, if the spacing of the stiffener is small, as compared to the in-plane dimensions of the plate, then perhaps we could smear the stiffness properties of the stiffener over the skin, rather than modeling the stiffeners discretely
- The following are possible ways in which a stiffener might be modeled discretely:
  - Beam elements
  - Shell elements (both conventional and continuum)
  - Solid elements

Abaqus Usage (2/6)

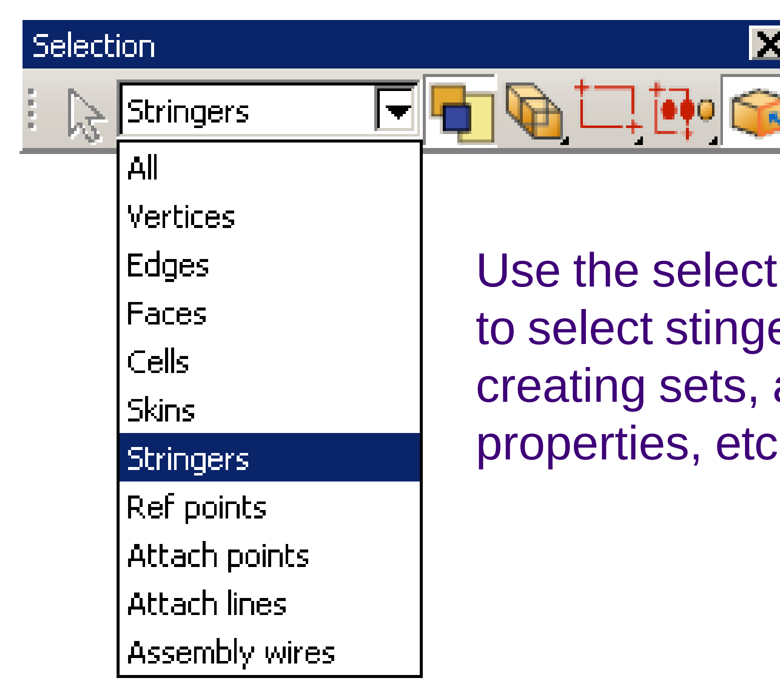
- Using beam elements to model a stiffener
  - Beam elements might be utilized as a cost-effective way to model the stiffeners accurately.
  - Geometry must conform to the beam assumption that the cross-sectional dimensions are small compared to the length of the beam.
  - For isotropic materials, the material definition for the beam element is fairly straightforward.
  - A composite stiffener will, in general, be composed of different materials throughout the cross section.
    - The meshed beam cross-section technique mentioned in Lecture 3 may provide a solution to determining the section behavior in this instance.
  - Beam elements preclude the use of layers that may induce bending-twisting coupling (e.g., an unsymmetric laminate).
  - The stringer capability in Abaqus/CAE provides a convenient method for creating beam reinforcements.

Abaqus Usage (3/6)

- Beam reinforcement using stringers
  - A stringer reinforcement defines a stringer that is bonded to the edge of an existing part and specifies its engineering properties.
  - An edge of a three-dimensional solid part or the edge of a two-dimensional planar part can be a stringer.
    - Beam or truss elements that share nodes with the underlying elements are generated.
  - Stringer layers on a geometric edge will have the same tangent direction; stringers on wires will inherit the tangent direction from the underlying wire.



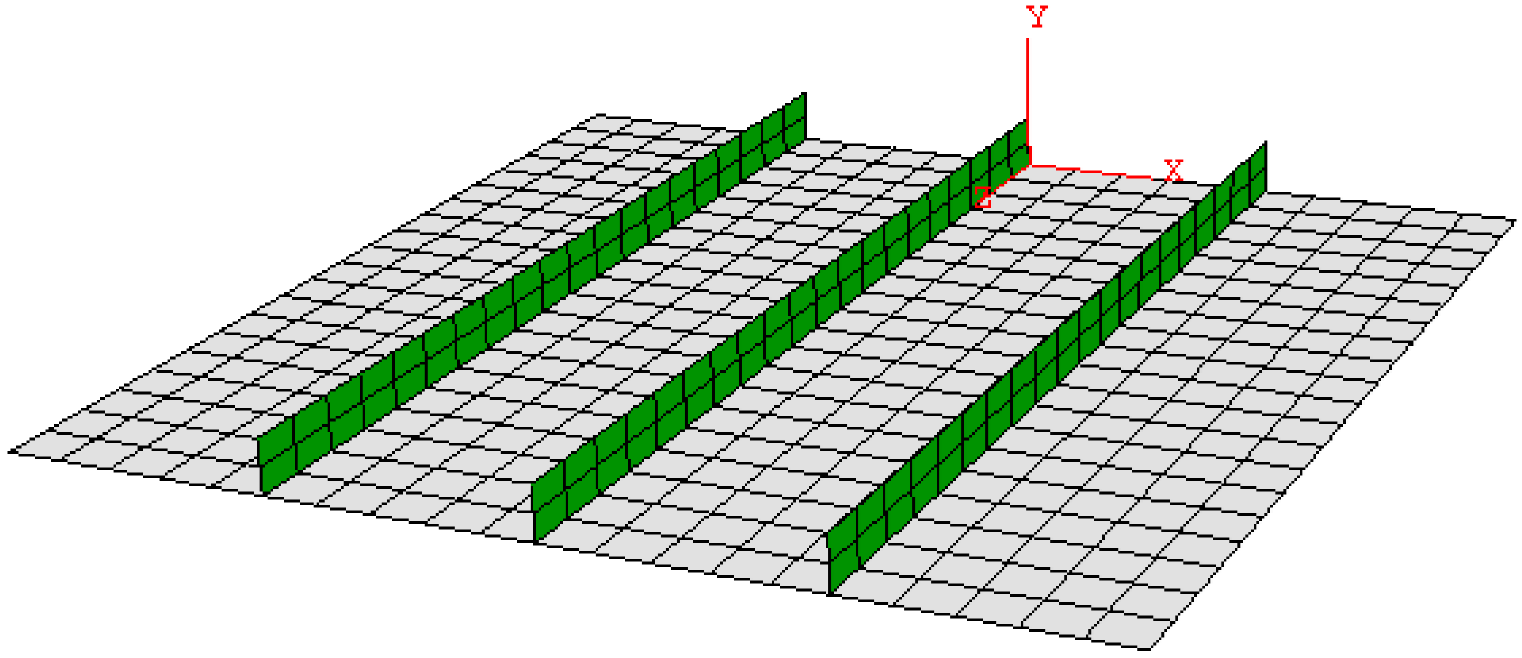
steel-reinforced beam



Use the selection options to select stingers when creating sets, applying properties, etc.

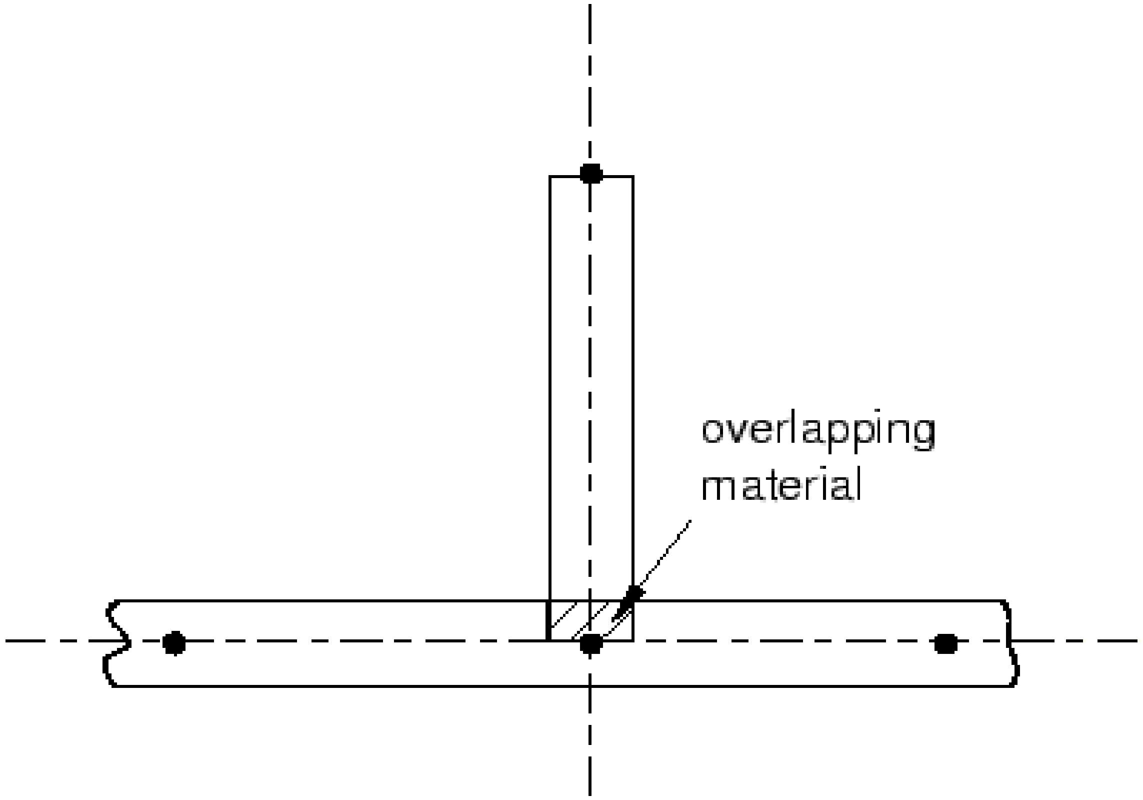
Abaqus Usage (4/6)

- Using shell elements to model a stiffener
  - Shell elements may be needed to model the stiffeners accurately if the restrictions regarding the utilization of beam elements are too great for your purposes
  - Again, utilization of shell elements requires that the geometry of the stiffener adhere to the underlying assumption of a shell:
    - The thickness dimension is much smaller than the other dimensions
  - This analysis methodology does allow for orientations of the lamina such that deformation couplings are possible (e.g., bending-twisting coupling)



Abaqus Usage (5/6)

- Shell elements allow you to offset the reference surface (location of the nodes) from the mid-surface
- This feature can be utilized to correct the material overlap issue that can occur when two shell elements are attached perpendicularly to each other



Abaqus Usage (6/6)

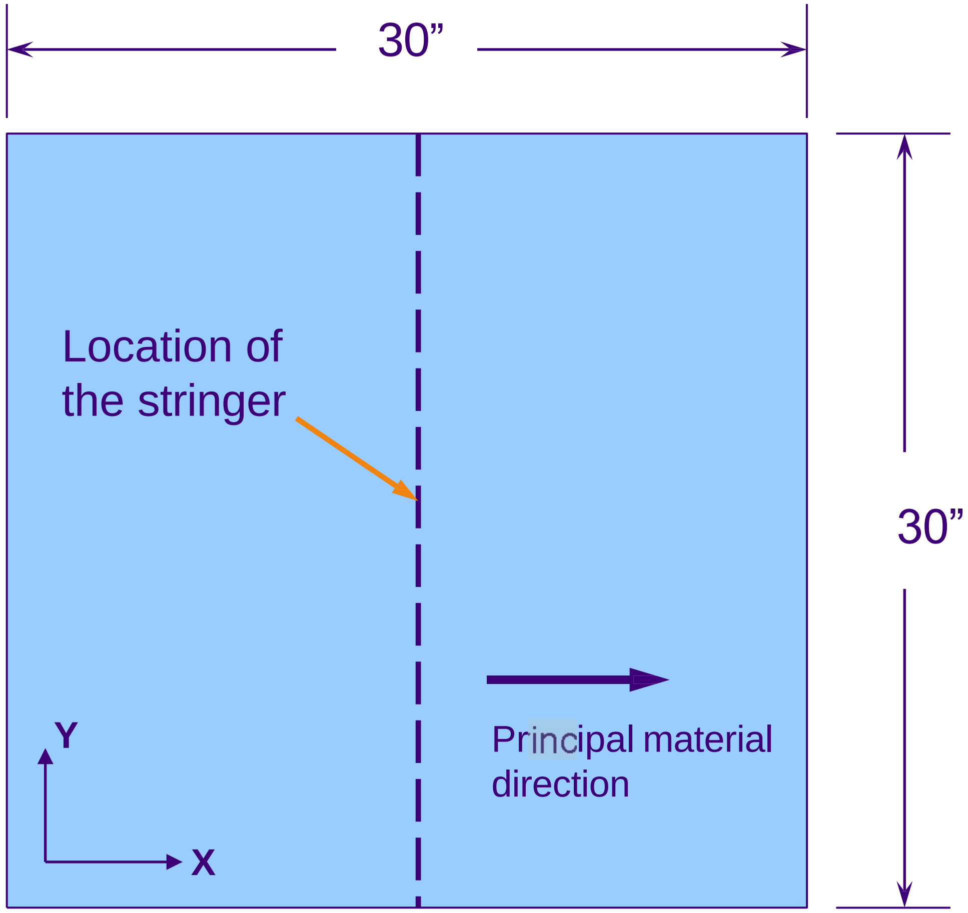
- Using solid elements to model a stiffener
- Utilization of solid elements represents the most general way to model the stiffeners, as well as the most computationally expensive (3D discretization)
- Local refinement can be plished if you utilize solid elements in the area of interest, and some other type of element globally
- The two regions would be coupled together utilizing one of Abaqus' coupling constraints (shell-to-solid, kinematic or distributing coupling, etc.)

Abaqus Example (1/9)

- Composite Laminate with Single Stiffener
  - A composite plate is analyzed with a single stringer to act as a bracing element
  - The stringer itself has the profile of a T-joint, which is adhesively bonded to the skin
  - All four edges of the plate are simply supported, and a uniform pressure load of 10 psi is applied to the face of the skin
  - The stiffener is assumed isotropic with $E = 1e7$ psi and $= 0.3$
  - The following material properties are utilized for the skin:

$$E_1 = 1.0 \times 10^7 \text{ psi}, E_2 = 4.0 \times 10^6 \text{ psi}, {}_{12} = 0.3,$$

$$G_{12} = G_{13} = G_{23} = 1.875 \times 10^6 \text{ psi}$$



The thickness of the skin is 0.25”