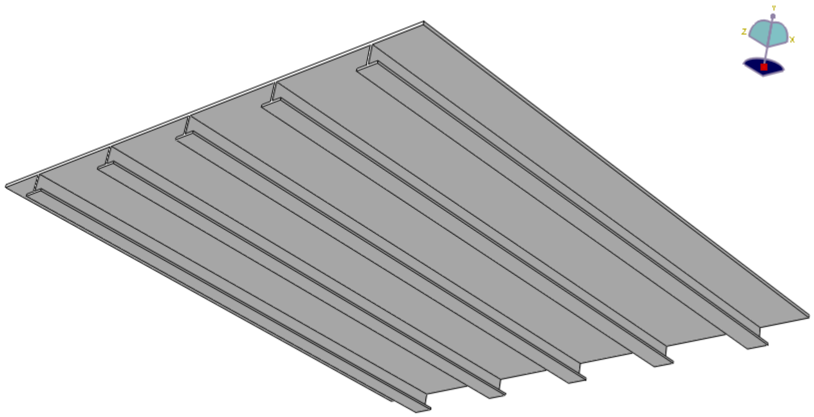
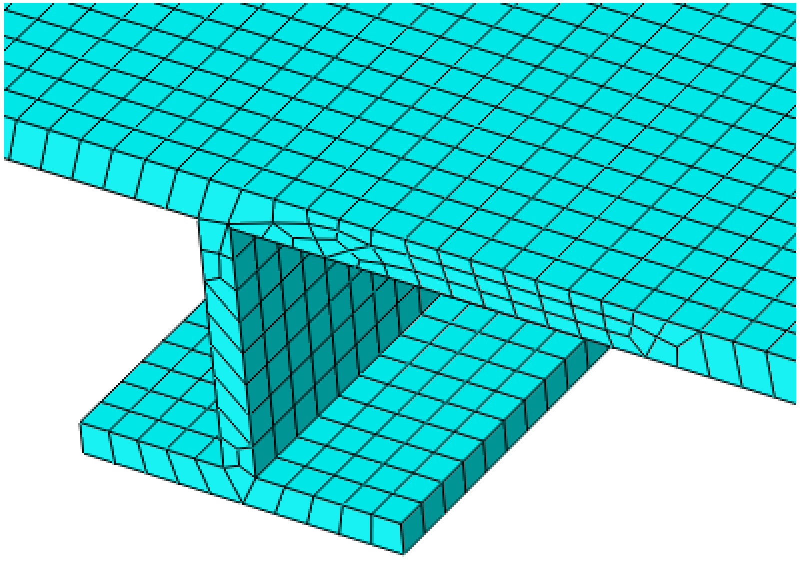
Tutorial: Stiffened Pane

Solid Finite Element for Thin Panel

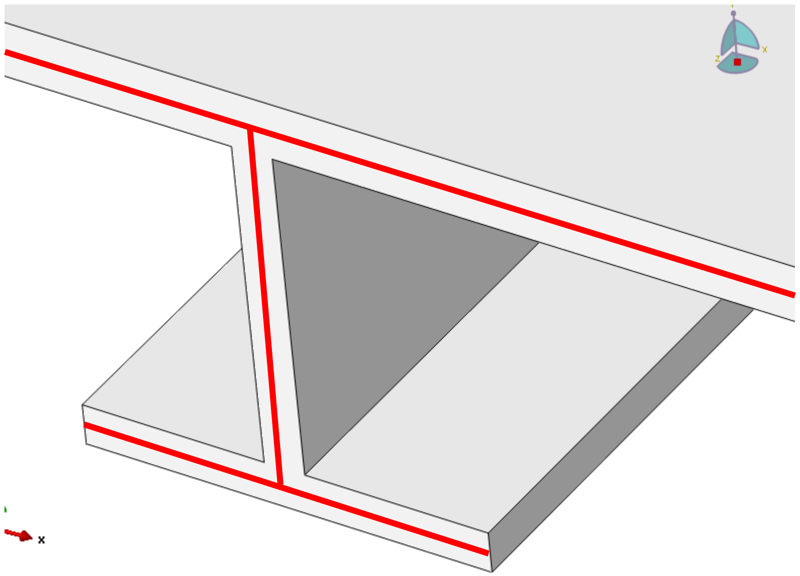
- Many mechanical systems are composed of thin plate-type panels
- Thickness dimension is much smaller than other dimensions
- Challenge in meshing
- At least 4 – 5 elements in thickness in order to estimate bending accurately





Shell Finite Element

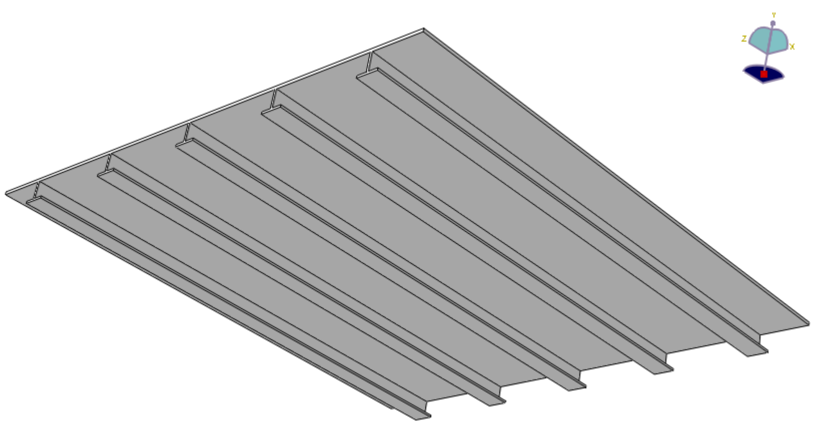
- Instead of making thin plate-like structure using solid elements, we take the neutral plane and analytically integrated over the thickness
- Model only mid-plane and provide thickness of each plane
- Shell elements on the mid plane
- In general, shell is more accurate than solid for thin panel, especially when bending is dominant



Stiffened Panel

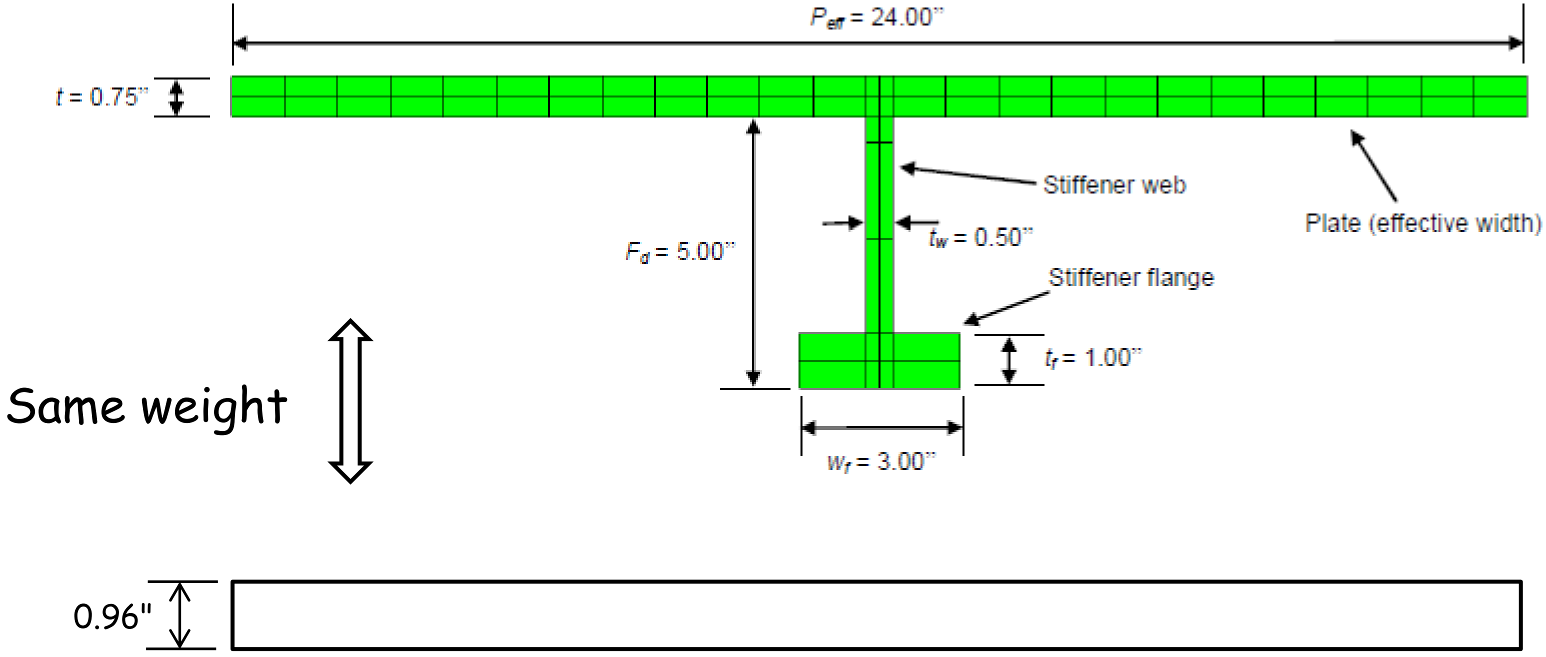
- In automotive industry, sheet stamping process is used to bend flat panel to curved shape in order to increase the flexural rigidity (EI)

- In aircraft structure, stringers and stiffeners are used to increase EI



Stiffened vs. un-stiffened panels

- We will try to compare the bending efficiency of stiffened versus un-stiffened panel with the same weight
- Assume that 5 of the following segment are connected

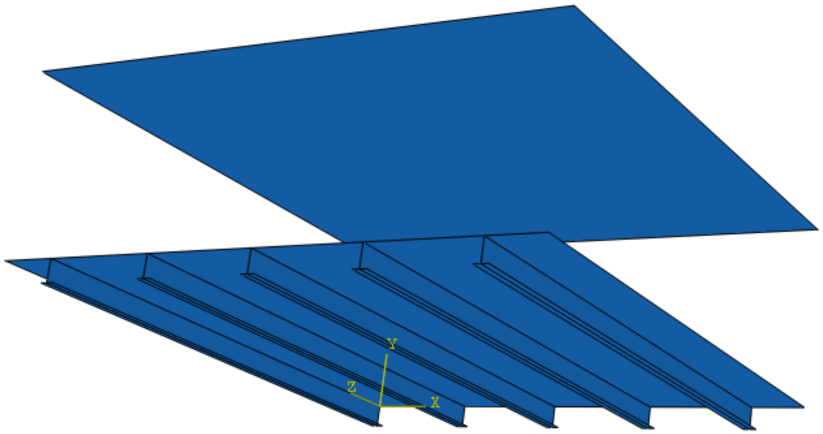


Create Parts

- Create a part with mid-plane
  - Part -> Name: stiffened -> 3D -> Deformable -> Shell -> extrusion
- Sketch line of one segment
  - (-12,0)–(0,0)–(12,0), (0,0)–(0,-4.875), (-1.5,-4.875)–(1.5,-4.875)
- Copy the segment using pattern
  - Linear pattern -> choose all lines -> Direction 1: Number = 5, Spacing = 1, Direction 2: Number=1
- Extrude by 120"
- Create unstiffened panel (120"x120")
  - Part -> Name: unstiffened -> 3D -> Deformable -> Shell -> extrusion
- Sketch line of entire span
  - (0,0)–(60,0)–(120,0), Extrude by 120"

Material and panel thicknesses

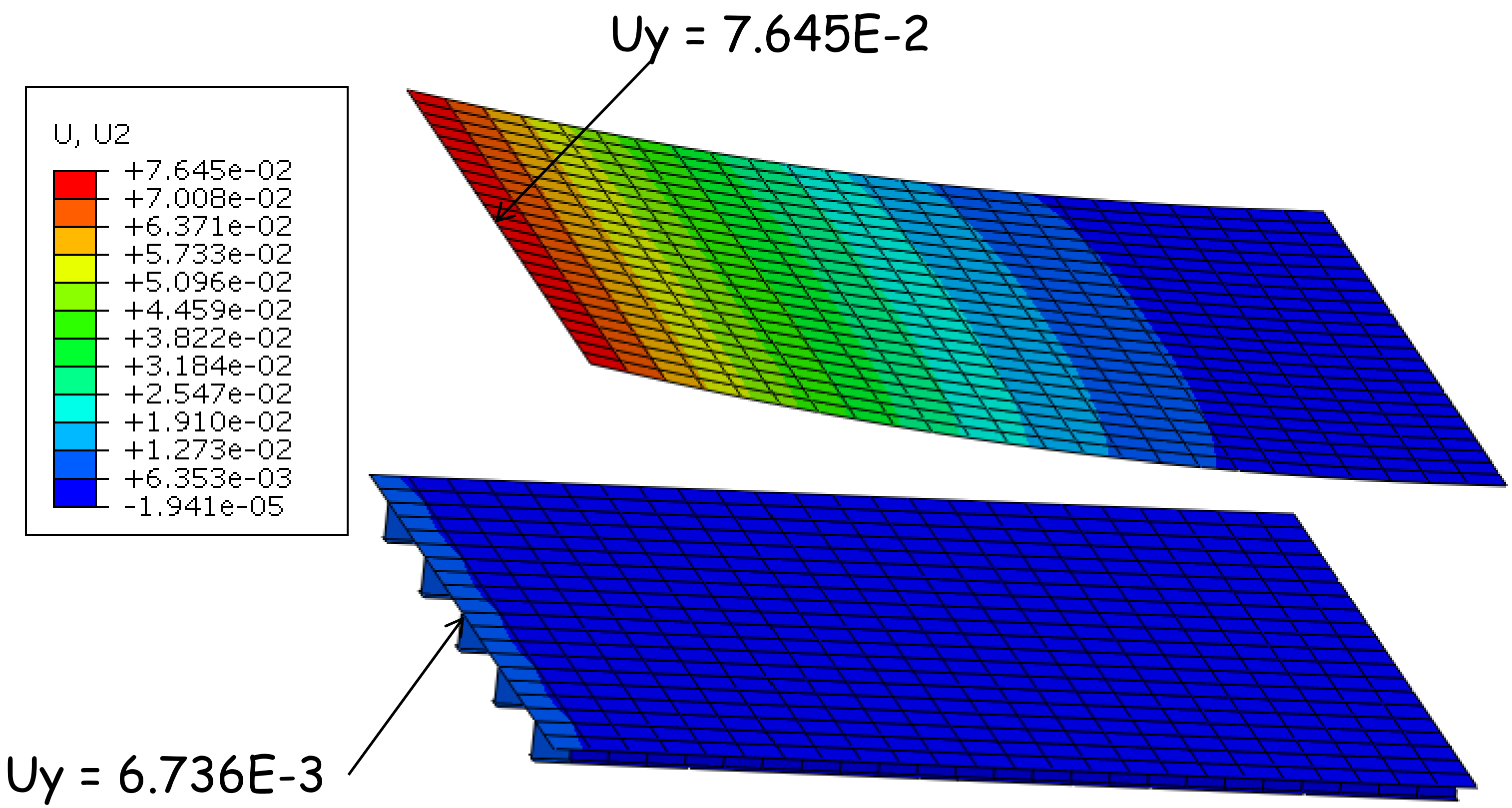
- Material property: E = 107 psa, NU = 0.3
- Section -> Shell -> Homogeneous
  - 0.75", 0.5", 1.0", 0.96"
- Make mesh with global element size = 5
- Assembly
  - Take two instances (stiffened, unstiffened)
  - Move unstiffened panel in y-direction by 40 to separate them



Boundary and Loading Conditions

- Create a step (linear perturbation) named Applied moment
- Fix all DOFs of left ends of both plates at Initial step
- Constraint -> MPC -> select middle point at right end -> select edges at right end -> Done -> Done
- Repeat for stiffened panel
- Apply moment at the middle point at right end
  - Load -> Moment -> select right middle point -> CM1 = -1000
  - Repeat for stiffened panel
- Create job and submit the job

Deformation comparison



Stress Comparison