

WORKSHOP 3: Static Analysis of Stiffened Plate

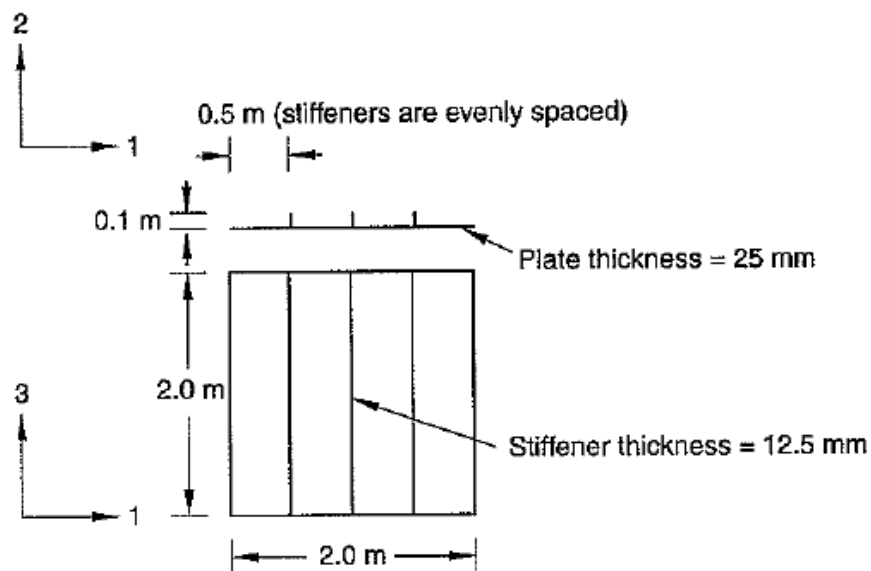
Preprocessing – Creating the model with ABAQUS/CAE

Defining the model geometry

Create a three dimensional, deformable part with an extruded shell base feature to represent the plate. Use an approximate part size of **5.0**, and name the part **Plate**. A suggested approach for creating the part geometry shown in figure 1.1 is summarized in following procedure:

To create the stiffened plate geometry:

1. To define the plate geometry, use **the create lines: connected** tool to sketch an arbitrary horizontal line.



Material properties

General properties:

$$\rho = 7800 \text{ kg/m}^3$$

Elastic properties:

$$E = 210 \times 10^9 \text{ Pa}$$

$$\nu = 0.3$$

Figure 1.1 Problem descriptions for static load on a flat plate

- To define the stiffener geometry, add three vertical lines extending up from the plate. The horizontal position of these lines is arbitrary at this stage, but their endpoints must snap to the horizontal line.
- Constrain the three vertical lines so they are equal length and dimension one of them so that it is 0.1 m long

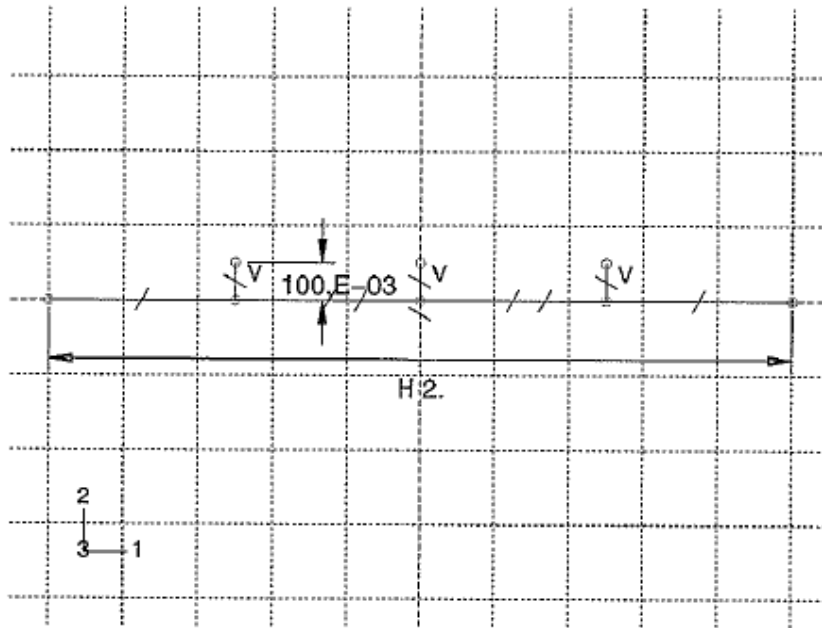


Figure 1.2 Sketch of the stiffened plate (1 out of 2 grid lines visible)

- Split the plate at the points where it intersects the stiffeners.
- Dimension the horizontal distance between the plate endpoints, and set the value to 2.0 m
- Apply equal length constraints to the horizontal segments of the line. The final part sketch is shown in figure 1.2
- Extrude the sketch to a depth of 2.0 m to create the plate

Defining the material properties

Define the material and section properties for the plate and the stiffeners. Create a material **Steel** with a mass density of **7800 kg/m³**, a young modulus of **210.0E9 Pa**, and a Poisson's ratio of **0.3**.

Creating and assigning section properties.

Create two homogeneous shell section properties, each referring to the steel material definition but specifying different shell thicknesses. Name the first shell section property **PlateSection**, select **Steel** as the material and specify **0.025 m** as the value for the **shell thickness**. Name the second shell section property **StiffSection**, select **steel** as the material, and specify **0.0125 m** as the value for the **shell thickness**.

Assign the **StiffSection** definition to the stiffeners (use [shift]+click to select multiple regions in the viewport).

Before assigning the **PlateSection** definition to the plate, consider the following. If the plate and the stiffeners are joined directly at their midsurfaces (this is default behaviour), an area of material overlap will occur, as shown in figure 1.3.

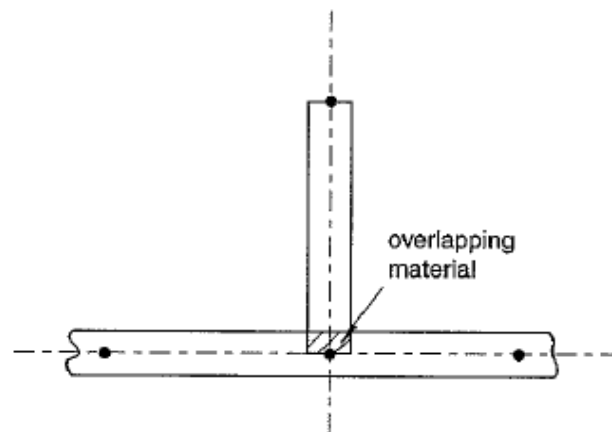


Figure 1.3 Overlapping material

Although the thickness of the plate and the stiffener are small in comparison to the overall dimensions of the structure (so that this overlapping material and the extra stiffness it creates would have little effect on the analysis results), a more precise model can be created by offsetting the plate reference surface from its midsurface. This technique allows the stiffeners to butt up against the plate without overlapping any material with the plate, as shown in Figure 1.4.

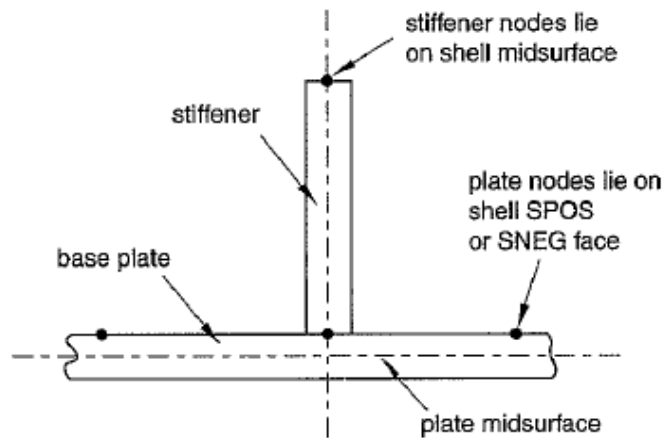


Figure 1.4. Stiffener joint in which the plate's reference surface is offset from its midsurface.

To determine whether to offset the plate reference surface to its positive (SPOS) or negative (SNEG) side, query the shell normals (**tools---query**) and note the color of the side of the plate facing the stiffeners (brown is the positive side; purple is the negative side). If necessary, flip the plate normals (**Assign---Normal**) so that its segments have consistent normals. Then assign the **PlateSection** definition to the regions of the plate. In the **Edit Section Assignment** dialog box, toggle on **Offset** and enter **0.5** as the value of the offset if the brown (positive) side of the plate faces the stiffeners and **-0.5** if the purple (negative) side faces the stiffeners.

Note: setting the offset value to **0.5** offsets the reference surface of the plate on half of the shell thickness away from its midsurface in the direction of the positive shell normal; setting the offset value to **-0.5** offsets the reference surface the same distance but in the opposite direction.

Creating an assembly

Create a dependent instance of the plate. Use the default rectangular coordinate system, with the plate lying in the 1-3 plane.

At this point, it is convenient to create the geometry sets that will be used to specify boundary conditions and output requests. Create one set named **edge** for the plate edges and one set named **Center** at the center of the intersection of the plate and the middle stiffener, as shown in figure 1.5. To create the set **Center**, you need to partition the edge of the original part in half using the **partition edge: Enter parameter** tool.

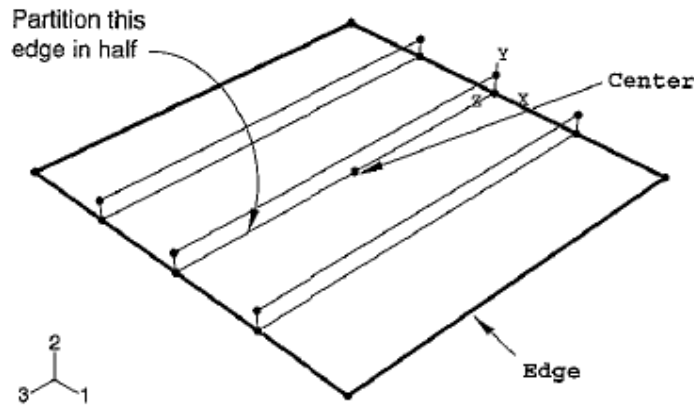


Figure 1.5 Geometry Set

Defining steps and output requests

Create a static, general step. Name the step **Static**, and specify the following step description: **Distributed pressure loading**. Enter a value of **1** for time period of the step. Create a history output request named **Center-U2** for the step **Static**. Select **Centre** as the output domain and select **U2** as the translation output variable.

Prescribing Boundary Conditions and Loads

Next, define the boundary conditions used in this analysis. Create a **Symmetry/Antisymmetry/Encastre** mechanical boundary condition in step **Static** named fix edges. Apply the boundary condition to the edges of the plate using the geometry set edge, and specify **ENCASTRE** ($U1=U2=U3=UR1=UR2=UR3=0$) to fully constraint the set.

The plate will be subjected to a pressure of 25000 Pa as shown in figure 1.6. Apply the pressure so that it pushes against the top of the plate (where the stiffeners are on the bottom of the plate). Such a pressure load will place the outer fibers of the stiffeners in tension

To define the pressure loading

1. In the model tree, double-click the **Loads** container. In the **Create Load** dialog box that appears, name the load **Pressure Load** and select **Static** as the step in which it will be applied. Select **mechanical** as the load category and pressure as the load type. Click continue
2. Select all the surfaces associated with the plate. When the appropriate surfaces are selected, click **done**. ABAQUS/CAE uses two different colors to indicate the two sides of the shell surface. To complete the load definition, the colors must be consistent on each side of the plate.

3. If necessary, select **flip surface** in the prompt area to flip the colors for a region of the plate. Repeat this procedure until all of the faces on the top of the plate are the same of the color.
4. In the prompt area, select the color representing the side of the plate without the stiffeners
5. In the **Edit Load** dialog box that appears, specify a uniform pressure of xxx Pa, select amplitude definition of **ramp**. Click **OK** to complete the load definition.

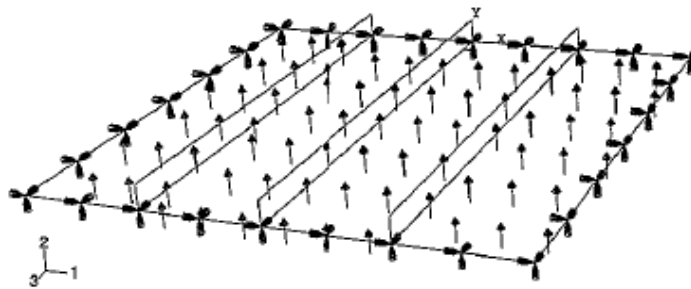


Figure 1.6. Pressure Load and Boundary Conditions

Creating the mesh and defining a job

Seed the part with a global element size of **0.1**. In addition, select **Seed—Edge by Number** and specify that two elements be created along the height of the stiffeners. Mesh element using quadrilateral shell element. The resulting mesh is shown in Figure 1.7. This course relatively coarse mesh provides moderate accuracy.

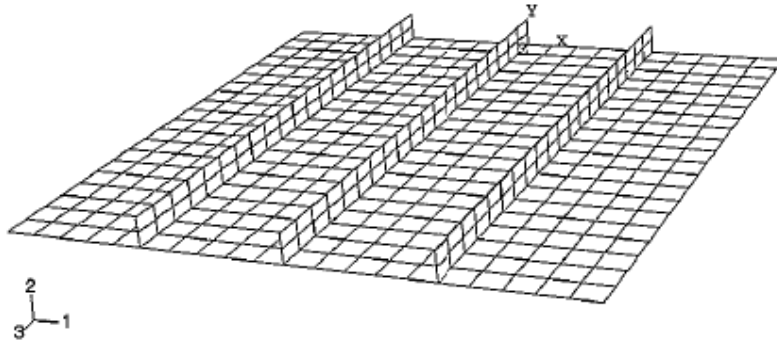


Figure 1.7. Meshed Plate

Create a job name **StaticLoad**.

When the job completes, enter visualization module, and open the .odb file created by this job. By default, ABAQUS plots the undeformed model shape with the shaded render style.

Postprocessing

History Output

Since it is not easy to see the deformation of the plate from the deformed plot, it is desirable to view the deflection response of the central node in the form of a graph. The displacement of the node in the center of the plate is of particular interest since the largest deflection occurs at this node.

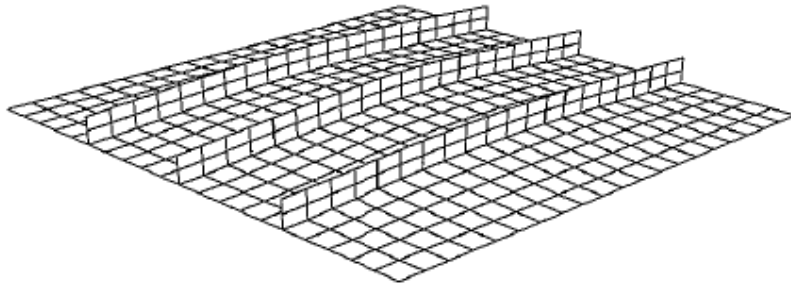


Figure 1.8. Displaced shape