Step-by-Step Abaqus Modeling of a 2D Shell Beam

This document provides a detailed, step-by-step guide for creating and analyzing a 2D shell beam model in Abaqus. It covers the entire workflow; including part creation, meshing, material definition, assembly, step setup, loading, boundary conditions, and output requests—using clear instructions, figures, and parameters for reproducibility.

1 Part

Create Part: Name = Part-1; 3D; Deformable; Shell; Extrude
Sketch: 0.089 (m) wide; See Figure [1]
Extrude: 0.0254 (m) depth; See Figure [1]

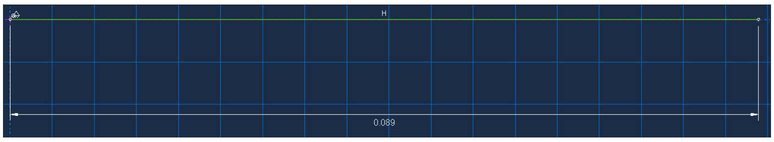

Figure 1

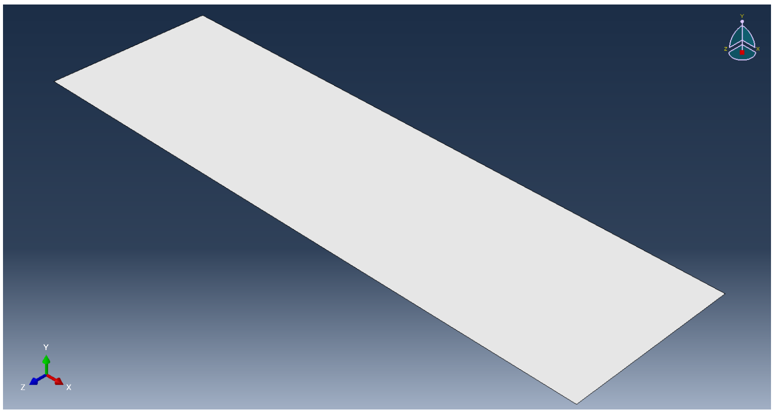

Figure 2

Create Partition: Face; Sketch; See Figures [3, 4]

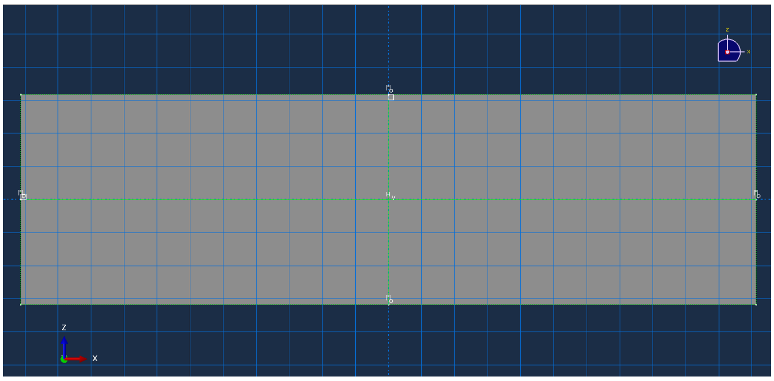

Figure 3: Draws lines halfway both long- and short-ways.

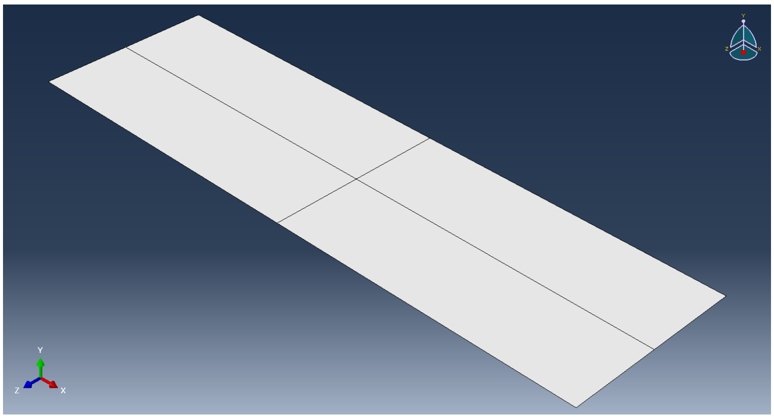

Figure 4: This creates a partition at the center of the part.

Create Set: Name = Set-1; See Figure [5]

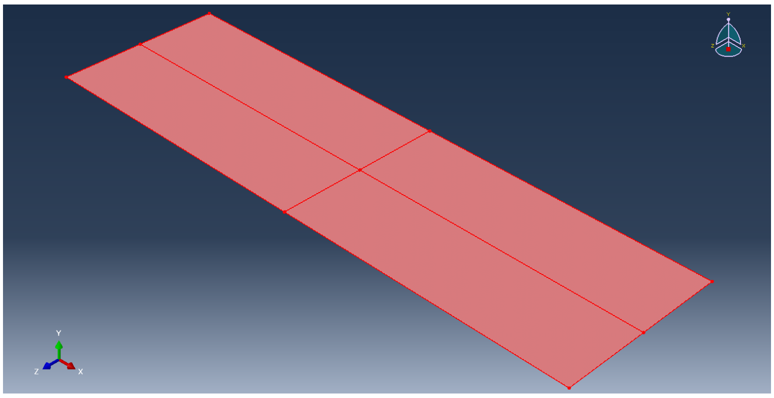

Figure 5: Select the entire body.

Create Set: Name = CenterNode; See Figure [6]

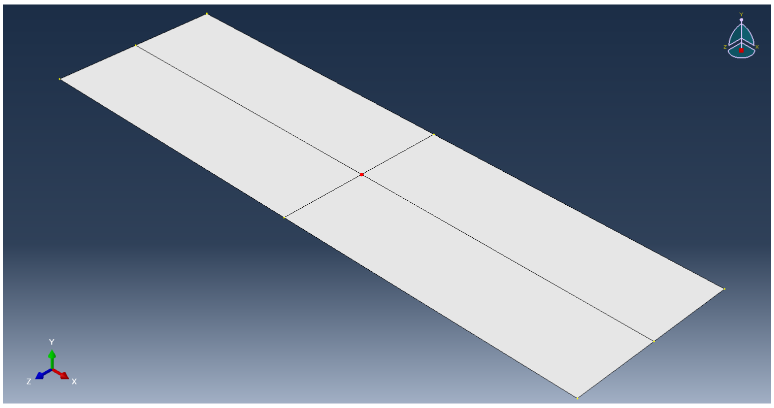

Figure 6: Select the point at the center of the created partition.

2 Mesh

Seed Part: Approximate global size = 0.002
Curvature control, Maximum deviation factor = 0.1
Minimum size control, By fraction of global size = 0.1

Select Mesh Part; See Figure [7]

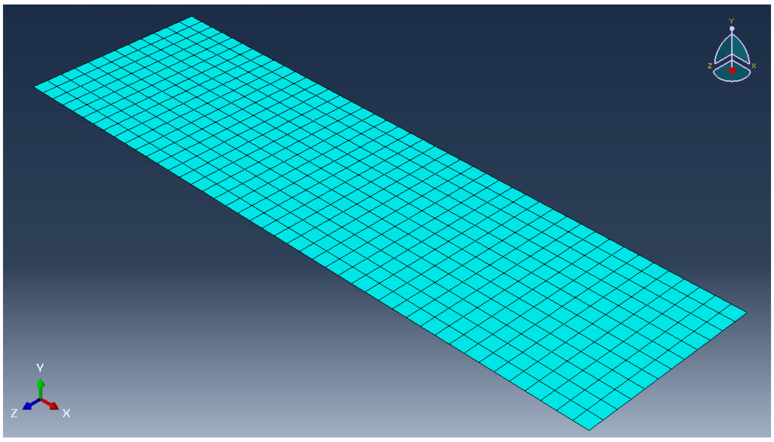

Figure 7

3 Material

Create Material: Name = FR4;
General, Density, Mass Density = 1900 (kg/m^3)
Mechanical, Elastic, Young's Modulus = 18.6e9; Poisson's Ratio = 0.136
Mechanical, Damping, Alpha = 65.53; Beta = 3.95e-6

4 Section

Create Section: Name = Section-2; Shell; Homogeneous
Section integration = During Analysis; Shell thickness, Value = 0.0016
Material = FR4; Thickness integration rule = Simpson; Thickness integration points = 5

5 Assembly

Create Instance: Auto; Parts; Parts = Part-1; See Figure [8]

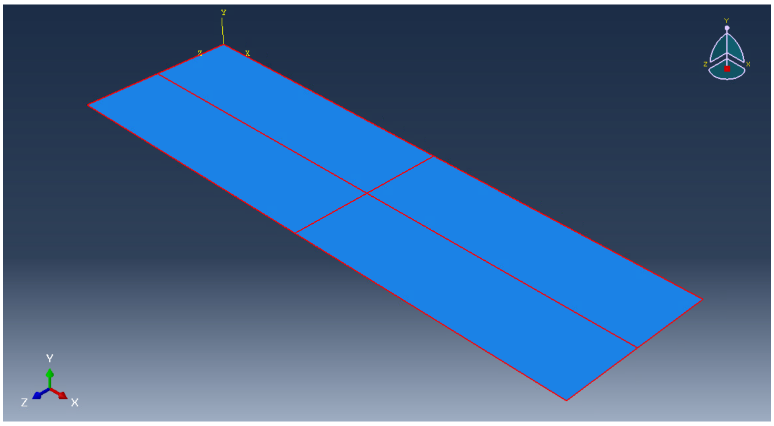

Figure 8

Create Set: Name = Set-1; See Figure [9]

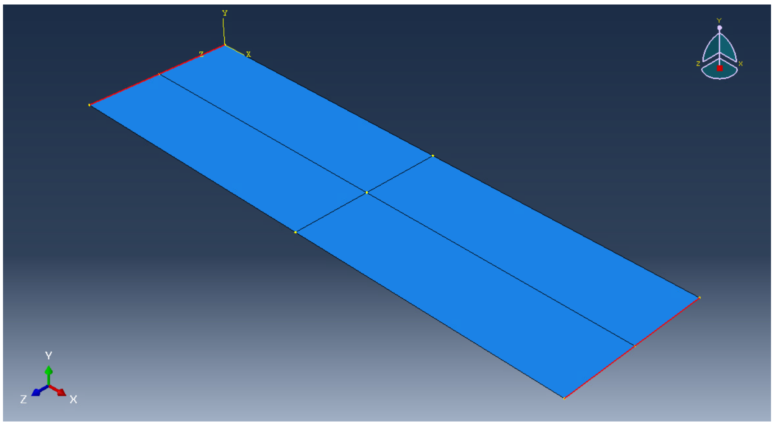

Figure 9: Select the the edges at the ends of the beam.

Create Set: Name = Set-2; See Figure [10]

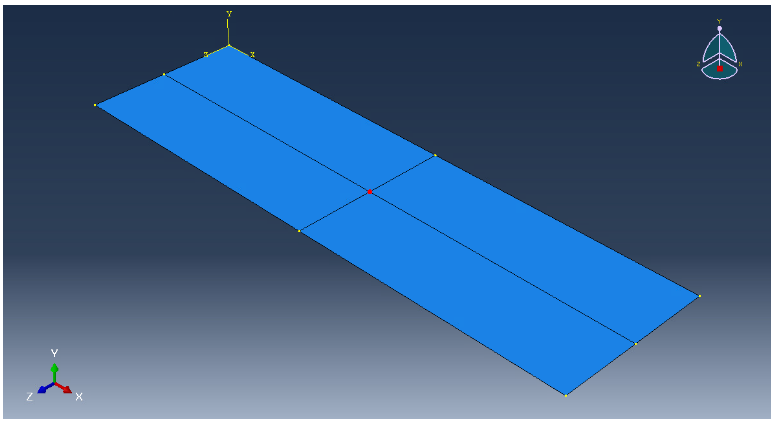

Figure 10: Select the point at the center.

6 Step

Create Step: Name = Force; Insert new step after Initial; General; Dynamic, Explicit
Under Basic: Time period = 0.05
Under Incrementation: Automatic; Global; Improved Dt Method; Unlimited; Time scaling factor = 1

7 Amplitude/Load

Create Amplitude: Tabular; Step time; Use solver default; See Table [1]

Table 1: Under Amplitude Data

	Time/Frequency	Amplitude
1	0	0
2	0.005	0
3	0.005001	1
4	0.0051	1
5	0.005101	0

Create Load: Name = Load-1; Step = Force; Mechanical; Concentrated Force; Region = Set-2
Distribution = Uniform; CF1 = 0; CF2 = -30; CF3 = 0; Amplitude = Amp-3; See Figure [11]

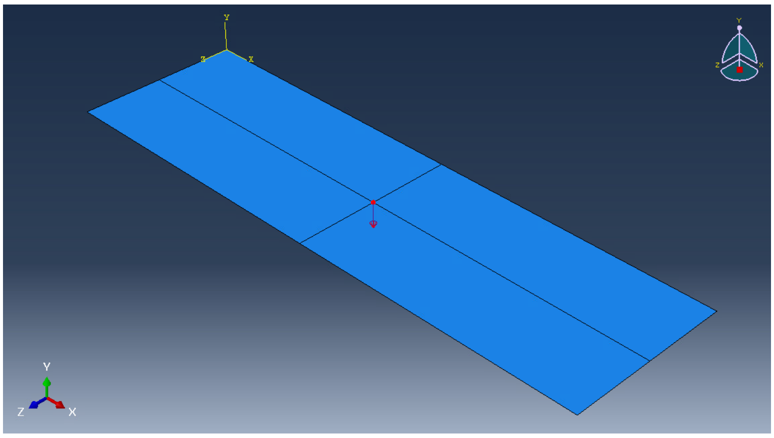

Figure 11

8 Boundary Conditions

Create BC: Name = FixedEnds; Step = Initial; Mechanical; Symmetry/Antisymmetry/Encaste
Select Set-1; ENCASTE (U1 = U2 = U3 = UR1 = UR2 = UR3 = 0); See Figure [12]

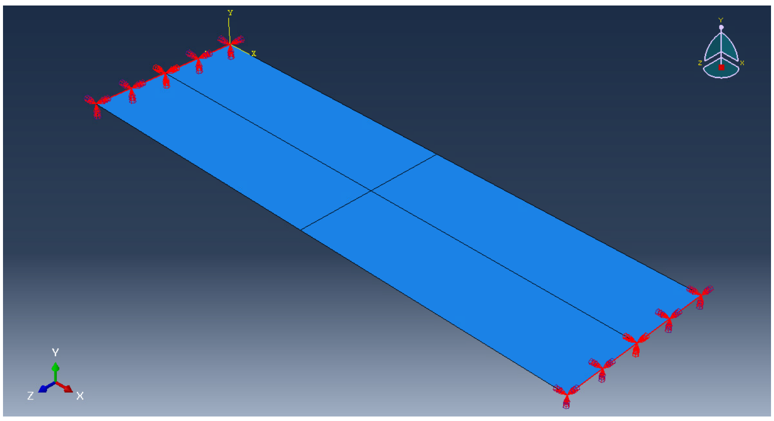

Figure 12

9 History Output Request

Create History: Name = CenterDisp; Step = Force
Domain = Set,Part-1-1.CenterNode; Frequency = Evenly spaced time intervals; Interval = 200

Under Output Variables: [Select from list below, Displacement/Velocity/Acceleration] U, Translations and rotations
Use defaults; Use global directions for vector-valued output

10 Job/Results

Create Job: Name = Job-1; Source = Model; Model-1
Submit Job; Once complete, select Job Results

Create XY Data: OBD history output; Spatial displacement, U2 at Node 1 in NSET CENTERNODE; See Figure [13]

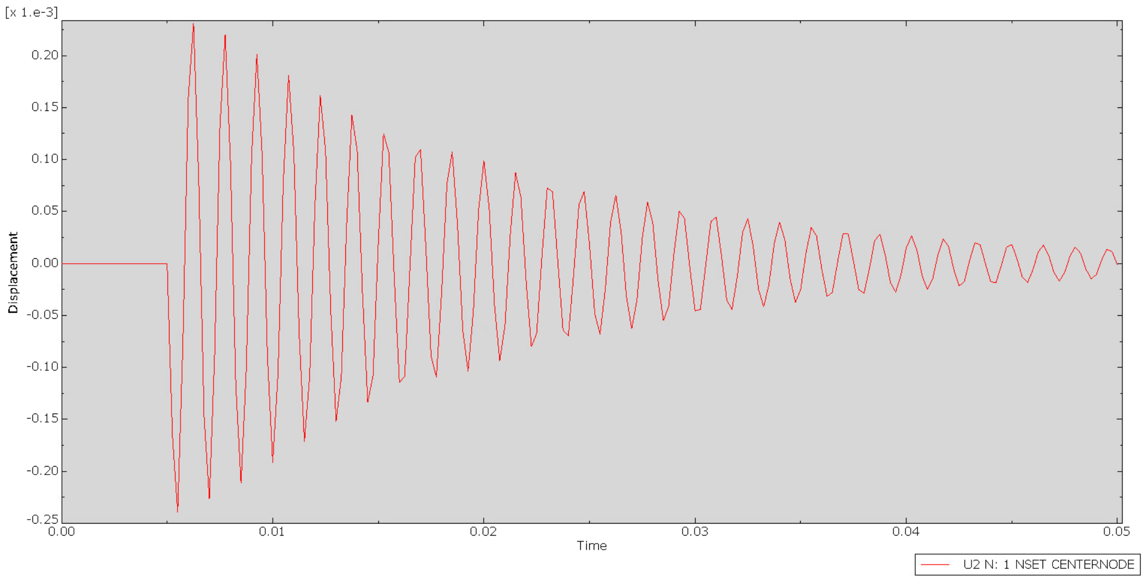

Figure 13: Time-series displacement of beam midpoint.