Problem Description

A table made of steel tubing with a solid steel top and shelf is loaded with an oblique impulse load. Determine the transient response of the table.
Analysis Steps

1. Start Abaqus and choose to create a new model database
2. In the model tree double click on the “Parts” node (or right click on “parts” and select Create)

3. In the Create Part dialog box (shown above) name the part and
   a. Select “3D”
   b. Select “Deformable”
   c. Select “Shell”
   d. Select “Extrusion”
   e. Set approximate size = 5
   f. Click “Continue...”

4. First, create the tabletop.
   a. Create the geometry shown below
   b. Extrude the top 1 meter.
5. In the toolbox area click on the “Create Datum Point: Offset From Point” icon.
   a. Select a corner of the tabletop, enter (0,-1,0) for the offset and press Enter.
   b. Repeat for the remaining 3 corners of the tabletop.

6. In the toolbox area click on the “Create Wire: Point to Point” icon
   a. Select “Disjoint wires” and “Merge wire with part geometry”, Click “Add...”
   b. Select a corner of the tabletop, then select the datum point directly below it.
   c. Repeat for the remaining three corners, Click “Done”, Click “OK”

7. In the toolbox area click on the “Offset Faces” icon
   a. Select the top of the table, Click “Done”
   b. Under Offset, select “Distance” and set the value to -0.5, Click “OK”

8. Double click on the “Stringers” node in the model tree
a. Select the edges of the tabletop, Click “Done”
b. Select the edges of the shelf, Click “Done”

9. Double click on the “Materials” node in the model tree

a. Name the new material and give it a description
b. Click on the “Mechanical” tab ➔ Elasticity ➔ Elastic
c. Define Young’s Modulus and Poisson’s Ratio (use SI units)

d. Click on the “General” tab ➔ Density
e. Density = 7800
f. Click “OK”

10. Double click on the “Profiles” node in the model tree
   a. Name the profile and select “Box” for the shape
   b. Click “Continue…”
   c. Enter the values for the profile of a=b=0.05 and a uniform thickness of 0.003 as shown below.
   d. Click “OK”

   ![Create Profile Window](image)
   ![Edit Profile Window](image)

11. Double click on the “Sections” node in the model tree
   a. Name the section “structural_tube” and select “Beam” for both the category and the type
   b. Click “Continue…”
   c. Leave the section integration set to “During Analysis”
   d. Select the profile created above (square_tube)
   e. Select the material created above (Steel)
   f. Click “OK”

   ![Edit Beam Section Window](image)

12. Double click on the “Sections” node in the model tree
   a. Name the section “plate” and select “Shell” Category and “Homogeneous” for Type
   b. Click “Continue…”

   ![Create Section Window](image)
c. Set the shell thickness to 0.006 and the material to the material created above.
d. Click “OK”

13. Expand the “Parts” node in the model tree, expand the node of the part just created, and double click on “Section Assignments”
   a. Select the legs
   b. Select the beam section created above (structural_tube)
c. Click “OK”
d. Select the stringers created along the shell edges
e. Select the beam section created above (structural_tube)
f. Click “OK”

g. Select the tabletop and shelf
h. Select the shell section created above (plate)
i. Click “OK”

14. Expand the “Assembly” node in the model tree and then double click on “Instances”
   a. Select “Dependent” for the instance type
   b. Click “OK”
15. Double click on the “Steps” node in the model tree
   a. Name the step, set the procedure to “Linear perturbation”, and select “Frequency”
   b. Click “Continue…”
   c. Give the step a description
   d. Number of eigenvalues requested = 50
   e. Click “OK”
16. Double click on the “BCs” node in the model tree
   a. Name the boundary conditioned “fixed_legs” and select “Symmetry/Antisymmetry/Encastre” for the type
   b. Click “Continue…”
   c. Select the lower end of all the legs and press “Done” in the prompt area
   d. Select “Encastre”.
   e. Click “OK”
17. Double click on the “Steps” node in the model tree
   a. Name the step, set the procedure to “Linear perturbation”, and select “Modal dynamics”
   b. Click “Continue…”
   c. On the basic tab, set the time period to 0.5 and the time increment to 0.0025 (total of 200 increments)
   d. On the damping tab, apply a critical damping fraction of 0.05 to all 50 modes
   e. Click “OK”

18. In the model tree double click on “Amplitudes”
   a. Select “Smooth step” and name the amplitude
   b. Enter the time and amplitudes shown below
   c. Click “OK”
19. Double click on the “Loads” node in the model tree
   a. Select “Line load” and click Continue...
   b. Select the short edge of the shelf
   c. Set Component 1 and 2 to -100000 (-100e3)
   d. Choose the amplitude created above and click “OK”

20. In the model tree under the “Assembly” node, double click on “Sets”
   a. Name the set, select the “Geometry” option, and click Continue...
   b. Select the corner of the table top above where the load was applied.

21. Double click on the “History output requests” node in the model tree
    a. Name the history output, select the modal dynamics step, and click Continue...
    b. Set the domain to “Set” and select the set created above
    c. For the frequency choose to output every 1 increments
d. For the output select the U1, U2 and U3 displacements

e. Click OK

22. In the model tree double click on “Mesh” for the Table part, and in the toolbox area click on the “Assign Element Type” icon
   a. Select all the beam sections (legs and stringers)
   b. Select “Standard” for element type
   c. Select “Linear” for geometric order
   d. Select “Beam” for family
   e. Note that the name of the element (B31) and its description are given below the element controls
   f. Click “OK”

   g. Select the table top and shelf
   h. Select “Standard” for element type
   i. Select “Linear” for geometric order
   j. Select “Shell” for family
   k. Uncheck reduced integration, Click “OK”
23. In the toolbox area click on the “Seed Part” icon.

   a. Set Approximate global size to 0.1. Click “OK”

24. In the toolbox area click on the “Mesh Part” icon
   a. Click “Yes” in the prompt area
25. In the menu bar select View ➔ Part Display Options
   a. Check the Render beam profiles option
   b. Click “OK”

26. Change the Module to “Property”
   a. Click on the “Assign Beam Orientation” icon
   b. Select the leg geometry from the viewport, Click “Done”
   c. Accept the default value of the approximate n1 direction, Press Enter, Click “OK”
   d. Select the long edges of the top and shelf, Click “Done”
   e. Accept the default value of the approximate n1 direction, Press Enter, Click “OK”
   f. Select the short edges of the top and shelf, Click “Done”
   g. Accept the default value of the approximate n1 direction, Press Enter, Click “OK”

27. In the model tree double click on the “Job” node
   a. Name the job
   b. Click “Continue…”
   c. Give the job a description
   d. Click “OK”
28. In the model tree right click on the job just created and select “Submit”

a. While Abaqus is solving the problem right click on the job submitted, and select “Monitor”
b. In the Monitor window check that there are no errors or warnings
   i. If there are errors, investigate the cause(s) before resolving
   ii. If there are warnings, determine if the warnings are relevant, some warnings can be safely ignored

29. In the model tree right click on the submitted and successfully completed job, and select “Results”

30. Display the deformed contour overlaid with the undeformed geometry
   a. In the toolbox area click on the following icons
      i. “Plot Contours on Deformed Shape”
      ii. “Allow Multiple Plot States”
      iii. “Plot Undeformed Shape”
31. In the toolbox area click on the “Common Plot Options” icon
   a. Note that the Deformation Scale Factor can be set on the “Basic” tab
   b. Click “OK”

32. In the menu bar click on Results ➔ Active Steps/Frames
   a. Uncheck the Extract Frequencies step
   b. Click “OK”

33. Click on the arrows on the context bar to change the time step being displayed
   a. Click on the three squares to bring up the frame selector slider bar
34. On the results tree, expand the History Output node and double click on the spatial displacement history created.