Buckling of a Simply-Supported Square Plate under Uniaxial Compression

A rectangular plate is simply-supported around its perimeter and is compressed uniaxially in the x direction.

The exact solution for the buckling load is given by

\[ P_{mn} = \pi^2 D \left( \frac{a}{m} \right)^2 \left[ \left( \frac{m}{a} \right)^2 + \left( \frac{n}{b} \right)^2 \right]^2 \]

where

\[ D = \frac{E t^3}{12(1-\nu^2)} \]

and m and n are integers (m=1,2,3, ...) and n=1,2,3, ...) which correspond to the number of sine waves in the x and y directions for the buckled mode shape given by

\[ w(x,y) = \sin \left( \frac{m\pi x}{a} \right) \sin \left( \frac{n\pi y}{b} \right) \]

For a prescribed uniform displacement in the x direction, the relationship between displacement and buckling load is given by

\[ \delta_{mn} = \frac{P_{mn} a}{Et} \]

Perform a finite element analysis for the following two cases:

- Case 1: 10 x 10 mesh using Q4 plate elements
- Case 2: 20 x 20 mesh using Q4 plate elements
Use the following properties:

- $\delta = 1$, $E = 30000$, $\nu = 0.3$, $a = b = 100$, $t = 1$

For both cases, extract 20 buckling modes then sort the lowest ten from lowest to highest based on the exact solution as given in the following table

<table>
<thead>
<tr>
<th>Mode number</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
<th>9</th>
<th>10</th>
</tr>
</thead>
<tbody>
<tr>
<td>$m$</td>
<td>1</td>
<td>2</td>
<td>3</td>
<td>2</td>
<td>4</td>
<td>3</td>
<td>1</td>
<td>4</td>
<td>5</td>
<td>5</td>
</tr>
<tr>
<td>$n$</td>
<td>1</td>
<td>1</td>
<td>1</td>
<td>2</td>
<td>1</td>
<td>2</td>
<td>2</td>
<td>2</td>
<td>1</td>
<td>2</td>
</tr>
</tbody>
</table>

Submit the following:
- Plot of the lowest ten eigenvalues vs mode number for exact, case 1 and case 2
- Plot of percent error in eigenvalues vs mode number for case 1 and case 2

The percent error is defined as

$$\% \text{ error} = \frac{\left| \delta_{mn}^{FE} - \delta_{mn}^{exact} \right|}{\left| \delta_{mn}^{exact} \right|}$$
ABAQUS/CAE Tutorial for FE Project 5

Module = Part
- Select “Create Part”, select “3D-Deformable-Shell-Planar”
- Create a rectangular part with corner coordinates of (0,0) and (100,100)

Module = Property
- Select “Create Material” and input the material properties
- Select “Create Section” and use “Shell – Homogeneous” then “Continue” and enter the “thickness”
- Select “Assign Section” and assign the section to the square region

Module = Assembly
- Select “Instance Part”, “Part-1”, “Independent (mesh on instance)” and “Continue”

Module = Step
- Select “Create Step” and “Procedure Type = Linear perturbation” and “Buckle” then “Continue”
- Set the “Number of eigenvalues requested” to 20

Module = Load
We will compress the plate by displacing the edge uniformly in the x direction (BC 4). We will apply the following four boundary conditions shown
BC 1
- Select “Create Boundary Condition” and “Displacement/Rotation”
- Select all edges around the plate perimeter by holding the Shift key and clicking them
- Set U3=0 around the perimeter with “Stress perturbation and buckling mode calculation”

BC 2
- Select “Create Boundary Condition” and “Displacement/Rotation”
- Select the left vertical edge
- Set U1=0 with “Stress perturbation and buckling mode calculation”

BC 3
- Select “Create Boundary Condition” and “Displacement/Rotation”
- Select the bottom horizontal edge
- Set U2=0 with “Stress perturbation and buckling mode calculation”

BC 4
- Select “Create Boundary Condition” and “Displacement/Rotation”
- Select the right vertical edge
- Set U1=-1 with “Stress perturbation only”

Module = Mesh
- Select “Seed Part Instance”, use a “Global element size” of 10 for Case1 and 5 for Case 2
- Select “Assign Element Type” and use defaults corresponding to the S4R element
- Select “Mesh Part Instance” and mesh the part

Module = Job
- Select “Create Job” and follow the usual procedure
- Select “Job Manager” and “Submit” the job

Module = Visualization
- Open the odb file
- Select “View”, “View Toolbox” and the 1-2 axes to get a front view

View contour plots to identify each mode (i.e. the m and n values)
- Select “Plot Contours”, then “Results” and “Field Output”
- Select U3 for transverse displacement
- Use the U3 contour plot to identify the m and n values
- Use the ODB frame buttons at the bottom to advance through each mode

Results
- Open the Job-1.msg file in a text editor
- Scroll down to the last iteration and copy the table of the converged eigenvalues