ABAQUS Tutorial – Column Buckling

Consider a 5 m column with a 10 cm circular cross-section (R=.05m) loaded in axial compression. The column is pinned at its ends. Determine the critical buckling modes and corresponding mode shapes

Theoretical Solution

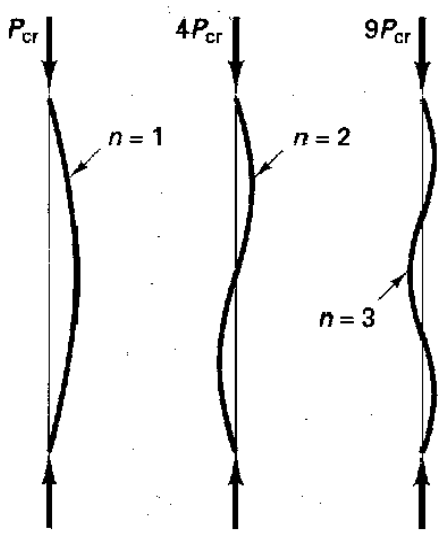
The theoretical Euler buckling loads are given by

$$P_{cr} = \frac{n^2 \pi^2 E I}{L^2}$$

For a steel column (E = 200 GPa) with I = 4.909e-6 m^4, the critical buckling loads and mode shapes are given by

Table 1. Theoretical Buckling Loads

n	P_{cr}
1	3.876e5
2	1.550e6
3	3.488e6
4	6.202e6
5	9.690e6
6	1.395e7



First three mode shapes

Finite Element solution (ABAQUS)

Start => Programs => ABAQUS 6.7-1 => ABAQUS CAE
Select 'Create Model Database'
File => Save As => create directory for files

Module: Sketch
Sketch => Create
Add=> Point => enter coordinates (0,0), (0,5) => select 'red X'
Add => Line => Connected Line => select point at (0,0) with mouse, then (0,5) , right click => Cancel Procedure => Done

Module: Part
Part => Create => select 2D Planar, Deformable, Wire, Approx size 10 => Continue
Add => Sketch => select 'Sketch-1' => Done => Done

Module: Property
Material => Create => Name: Material-1, Mechanical, Elasticity, Elastic => set Young's modulus = 200e9, Poisson's ratio = 0.3 => OK
Profile => Create => Circular => r=.05 => OK
Section => Create => Name: Section-1, Beam, Beam => Continue => Profile Name: Profile-1 => Linear Properties => E=200e9, G=77e9 => OK => OK
Assign Section => select all elements by dragging mouse => Done => Section-1 => Done
Assign Beam Section Orientation => select full model => Done => n_1 direction = 0.0,0.0,-1.0 => Done

Module: Assembly
Instance => Create => Part-1 => OK

Module: Step

Step => Create => Name: Step-1, Linear Perturbation, Buckle => Continue => Number of Eigenvalues requested: 6 => OK

Module: Load
Load => Create => Name: Step-1, Step: Step 1, Mechanical, Concentrated Force => Continue => select point at (0,5) => Done => set CF 1 =0, CF 2 = -1 => OK
BC => Create => Name: BC-1, Step: Step-1, Mechanical, Displacement/Rotation => Continue => select point at (0,0) => Done => U1=U2=0
BC => Create => Name: BC-1, Step: Step-1, Mechanical, Displacement/Rotation => Continue => select point at (0,5) => Done => U2=0

Module: Mesh
Seed => Edge by Size => select full model by dragging mouse => Done => Element Size=.25 => press Enter => Done
Mesh => Element Type => select full model by dragging mouse => Done => Element Library: Standard, Geometric Order: Linear, Family: Beam, Cubic interpolation (B23)=> OK => Done
Mesh => Instance => OK to mesh the part Instance: Yes => Done

Module: Job
Job => Create => Name: Job-1, Model: Model-1 => Continue => Job Type: Full analysis, Run Mode: Background, Submit Time: Immediately => OK
Job => Manager => Submit => Job-1
Results

Module: Visualization

Result => Step.Frame => view Eigenvalues (Buckling Loads) - see Table 2 below

Plot => Deformed Shape
View => Graphics Options => Background Color => White

Ctrl-C to copy viewport to clipboard => Open MS Word Document => Ctrl-V to paste image

Plot=> Contours => Result => Field Output => select S, Max. Principal => Section Points =>
Category: 'beam general' => select section points at +/- 2.5 to view stress contours.
Ctrl-C to copy viewport to clipboard => Open MS Word Document => Ctrl-V to paste image

Report => Field Output => Setup => Number of Significant Digits => 6
Report => Field Output => Variable => Position: Unique Nodal => select U: Spatial
Displacements, UR3: Rotational Displacements, S: Max. Principal => Apply
Cut and paste tabulated results from 'abaqus.rpt' file to MS Word document.

Table 2. Buckling Loads (FEA)

1	Mode 1: EigenValue = 3.87579E+05
2	Mode 2: EigenValue = 1.55033E+06
3	Mode 3: EigenValue = 3.48844E+06
4	Mode 4: EigenValue = 6.20257E+06
5	Mode 5: EigenValue = 9.69442E+06
6	Mode 6: EigenValue = 1.39674E+07

Buckled Mode Shapes: