Tutorial 3:

Plane Beam

CLAMPED-CLAMPED BEAM

- E = 100 MPa,
- Circular section with r1 = 20mm, r2 = 15mm
- Plot bending moment and shear force diagrams



ABAQUS BEAM ELEMENT

- Classification of Beam Elements
 - Dimension (2D / 3D)
 - Formulation type (Mindlin-2node / Mindlin-3node / Euler) Mindlin (Shear flexible), Euler (Cubic formulation)
 - Additional information (OS (open section) / H (hybrid) / OSH)



2D VS 3D BEAM ELEMENTS

- Degree of freedoms for a beam element
 - 2D/3D (2D beam is a special case of the 3D beam)
 - With translation (from the truss element)
 - Do not confuse sign convention for displacements and forces



GEOMETRY MODELING

- Parts
 - 2D Planar, Deformable, Wire, App Size = 4
 - Create lines: (-1, 0), (0,0), (1, 0)
- Materials
 - Mechanical, Elasticity, Elastic
 - Young's modulus = 100E6, Poisson's ratio = 0.3
- Profiles
 - Name: R20, Circular, Continue, r = 0.02, Ok
 - Name: R15, Circular, Continue, r = 0.015, Ok
- Sections
 - Name: R20, Beam, Beam, Continue, Profile name: R20, Ok
 - Name: R15, Beam, Beam, Continue, Profile name: R15, Ok

Section Assignment

- Part, Part-1, Section Assignments
- Select left beam, Done, Section: R20, Ok
- Section Assignments, Select right beam, Done, Section R15, Ok

SECTION ORIENTATION

- Beam cross section needs orientation (ex. Second moment of inertia)
- n₁ vector is (0, 0, -1) for plane beam (may not be modified for plane beam)
- Tangent direction vector t is dependent to the direction of wire geometry





SIGN CONVENTION

- Sign Convention for force and moment
 - Must not be confused with sign convention for displacements
 - Users must make sure which sign convention is used in a FEA software
 - Shear force and bending moment sign convention of ABAQUS for beam diagram



SIMULATION SETUP

- Assembly, Instance
- Steps
 - Linear perturbation, Static
- Field output request
 - Check SF, Section forces and moments
- BCs
 - Initial, Encastre (clamped)
- Loads
 - Mechanical, Pressure (force/length for beam) or Line load (force/length), select upward, Uniform, 30
- Mesh
 - Element type, "Classical beam theory" or Euler beam theory, Cubic polynomial, (B23), Global element size = 0.2.
 - Generate elements: Mesh/Parts



POSTPROCESSING

- Analysis, Create Job, Data Check, Submit
- Results
- Deformed plot, Stress plots
 - Result/Field output, Section points, Top or Bottom
 - View/ODB Displace Option/Render beam profiles



n2

nl

top

SHOWING ELEMENT NORMAL

- Common Plot Options
 - Normals, check "Show normals", "On element"

Common Plot Options					
Basic Color & Style Labels Normals Other					
Show normals On elements On surfaces					
Colors					
Face normal: 💻					
Beam tangent:					
Beam n1:					
Beam n2:					
Style					
Length: Medium					
Line thickness: Arrowhead:					
OK Apply Defaults Cancel					

- Module part, View, Part Display Options
 - check "Render Beam Profile"

Render Sty	
	/le
Wirefram	me 🔘 Hidden 💿 Shaded
Geometry	
Show d	otted lines in hidden render style
Show e	dges in shaded render style
Show si	lhouette edges
🔲 Highlig	ht only visible entities
Face highli	ghting: Stippling
Curve refin	ement: Coarse
Note: The to t	e refinement setting will be applied only he current part.
Mesh	
Show: Ext	erior edges
Show e	dges in shaded render style
🔲 Highlig	ht only visible entities
Idealizatio	ns
Render	beam profiles

BENDING MOMENT DIAGRAM

- Result/Field Output/SM
- Contour Plot Options/Show tick marks for line elements



- Shear forces (SF2) are not available for B23 element
- Paths, Node list
 (first node #, last node #, inc)

Edi Name Type: Dath	t Node List Path : Path-1 Node List	-		×
	Part Instance	Node Labels (Examples:	5 5:10 5:10:2)	-
1	PART-1-1	1:11:1		
View	port selections:	dd Before Add After		Ŧ
			Const	

BENDING MOMENT DIAGRAM cont.

XY Data from Path

XYData, path, X Values = Sequence ID

- Field output, SM1

	🖉 🗇 Data Extraction	
• •	💽 Create XY Data 📃 💌	Path: Path-1
💽 XY Data Manager	Source	Model shape: O Deformed Undeformed
Data Source	ODB history output	Point Locations:
Ourrent session Current ODB: Job-1.odb	ODB field output	Include intersections
Name Description	Thickness	
	Operate on XY data	X Values
	ASCII file	True distance X distance
	Keyboard	Normalized distance Y distance T
	Path	Sequence ID 🔘 Z distance
		Y Values
	Continue Cancel	Step: 1, Step-1
	· · · · · · · · · · · · · · · · · · ·	Frame: 1 Step/Frame
		Field output variable: SM, SM1 (Avg: 75%) Field Output.
		Note: Result option settings will be applied to calculate result values for the current step and frame.
		Save As Plot Cancel
2 4 4 4 1 4 1 4 1 4 1 4 1 4 1 4 1 4 1 4	SM, SM1 (Avg: 75%): Seq Id. along 'Path-1'	

- To adjust font size of the XYData plot
 - Double click object to open a dialog box to adjust its properties (ex: double click the legend box to enlarge its font size)

SHEAR FORCE IN TIMOSHENKO BEAM

- Change element type
 - Element type, "Timoshenko beam theory" or Mindlin beam theory, Shear flexible, (B21), Global element size = 0.2
- Field Output
 - Shear force output (SF2) is available



Buckling Tutorial

Buckling of Column

- In plane beam, we assume that axial deformation (bar) is independent of bending deformation (beam)
- This is normally true, but when the deformation is large or the force is large, the coupling effect become important





- Buckling: beam collapses under compressive axial load
- Critical load
 - When the compressive load is less than the critical load, the beam deforms as a bar
 - When the load reaches the critical load, the beam suddenly collapses

Load Factor

- For a given applied load F, each element
 has axial element force P^(e)
- · Load Factor
 - Let's assume that the structure is stable at the current load F
 - How much the load can increase before buckle?

$$P_{cr} = \lambda \cdot F \quad \longrightarrow \text{Load factor}$$

- Due to linearity, element forces will also be $\lambda \cdot P^{(e)}$
- All elements will have different $P^{(e)}$, but λ will be the same
- The goal is to find λ when the structure becomes unstable

Instability _____ Large (infinite) displacement



Finite Element Equation for Buckling

Stiffness + incremental stiffness

$$[\mathbf{K} - \lambda \mathbf{K}_{inc}]\{\mathbf{Q}\} = \{\mathbf{F}\}\$$

$$[\mathbf{k}_{inc}^{(e)}] = \frac{P^{(e)}}{30L} \begin{vmatrix} 36 & 3L & -36 & 3L \\ 3L & 4L^2 & -3L & -L^2 \\ -36 & -3L & 36 & -3L \\ 3L & -L^2 & -3L & 4L^2 \end{vmatrix} \begin{vmatrix} \mathbf{v}_i \\ \mathbf{\theta}_i \end{vmatrix}$$

Incremental stiffness

- The stiffness is reduced due to coupling effect of compressive load
 - $P_{cr} = \lambda \cdot F$

- λ : Load factor, eigenvalue
- $P^{(e)}$: Element force
- P_{cr} : Critical load

Finite Element Equation for Buckling

$$[\mathbf{K} - \lambda \mathbf{K}_{inc}]\{\mathbf{Q}\} = \{\mathbf{F}\}\$$

- This matrix equation has a small displacement $\{Q\}$ if $[K \lambda K_{inc}]$ is positive definite
- Instability occurs when the matrix is singular

 $|\mathbf{K} - \lambda \mathbf{K}_{inc}| = 0$

Among many eigenvalues, the lowest one will be used to calculate the critical load

$$P_{cr} = \lambda \cdot F$$

Buckling with Hinged Ends

- E = 20E6 psi, L = 200in, F = 1 lb, square section 0.5"X0.5"
- Theoretical buckling load = 25.7 lb



GEOMETRY MODELING

- Parts, 2D Planar, Deformable, Wire, Approximate size = 200
- Create lines, (0,100), (0,0), Done
- Materials, Mechanical, Elasticity, Elastic, Young's modulus
 = 2E7, Poisson's ratio=0.3, OK
- Profiles, Rectangular, a=0.5, b=0.5
- Sections, Beam,
 Profile name = Profile-1
 Material name = Material-1
- Part, Section Assignment, "Select the line", Done, Section = Section-1



MESHING

- Module = Property
- Assign beam orientation, 📽 "Select the line", Done, n1 direction = (0, 0, -1), OK
- Parts, Mesh
- Mesh, Element Type, "Select the line", Done, Standard, Beam, Linear, Cubic formulation, OK
- Seed, Edges, "Select the line", Done, By number, Number of elements = 10, OK
- Mesh, Part..., Yes
- View, Part Display Option, Mesh, Show node labels, Show element labels, OK

MESHING

- Assembly, Instance, Part-1, OK
- Steps, Linear perturbation, Buckle, Number of eigenvalues requested = 5
- BCs, Step=Initial, Displacement/Rotation, "Select top node", Done, U1, OK
- BCs, Step=Initial, Displacement/Rotation, "Select bottom node", Done, U2, UR3, OK
- Loads, Step-1, Concentrated force, "Select top node", Done, CF2=-1
- Jobs, Job-1, Continue..., OK
- "Right click on Job-1", Submit
- "Right click on Job-1", Results



F=1

000

U2=UR3=0

U1=0 ğ

POSTPROCESSING

Session Data

ė 🦰

Output Databases (1)

Job-1.odb

Steps (1)

Step-1

History Output

Frames (6)

Frame: 1

- Output Database, Job-1.odb, Steps, Step-1, Frames, "Select Frame:1"
- Plot Deformed Shape, "EigenValue = 25.702"
- Frame: 2 Frame: 3 Repeat for other frames Frame: 4 Frame: 5 EigenValue=25.702 EigenValue=642.88 EigenValue=2092.4 EigenValue=231.33 EigenValue=1261.8 ard 3DEXPERIENCE R2019x EXPERIENCE R2019x ard 3DEXPERENCE R2019x ard 3DEXPERENCE R2019x ard 3DEXPERIENCE R2019x 92.4 1.33 2.88 51.8 actor: +1.000e+01 cale Factor: +1.000e+01 cale Factor: +1.000e+01 cale Factor: +1.000e+01 cale Factor: +1.000e+01

MIRROR IMAGE

 View, ODB Display Options, Mirror/Pattern, Mirror plane = XZ



UNDERSTAND BUCKLING LOAD

Eigen	Buckling load	
F=1 lb	F=10 lb	P _{cr}
25.702	2.5702	25.702
231.33	23.13	231.33
642.88	64.28	642.88
1261.8	126.18	1261.8
2092.4	209.24	2092.4

Buckling Load = Applied force * Eigenvalue

The lowest buckling load is the most important!!