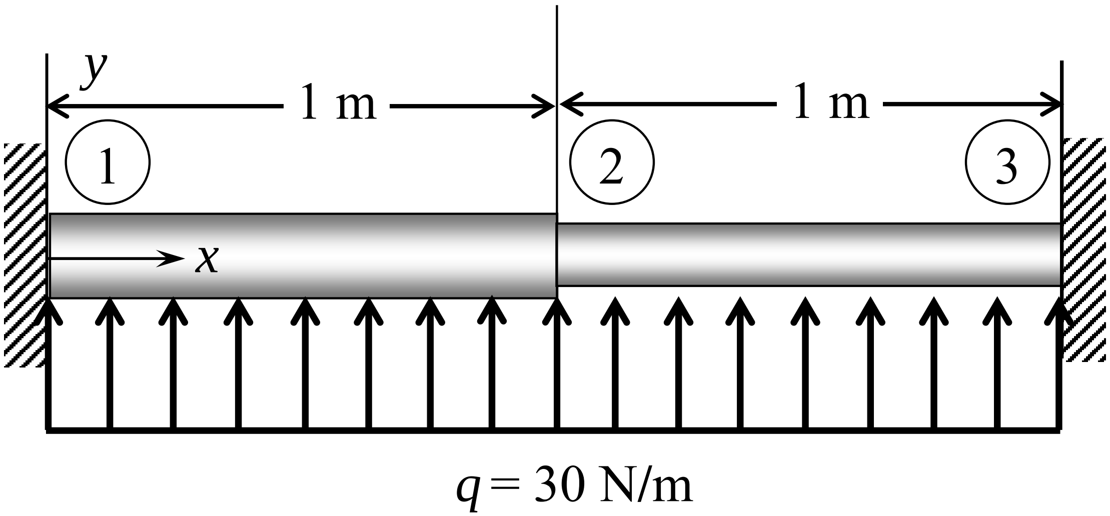
Tutorial 3:

Plane Beam

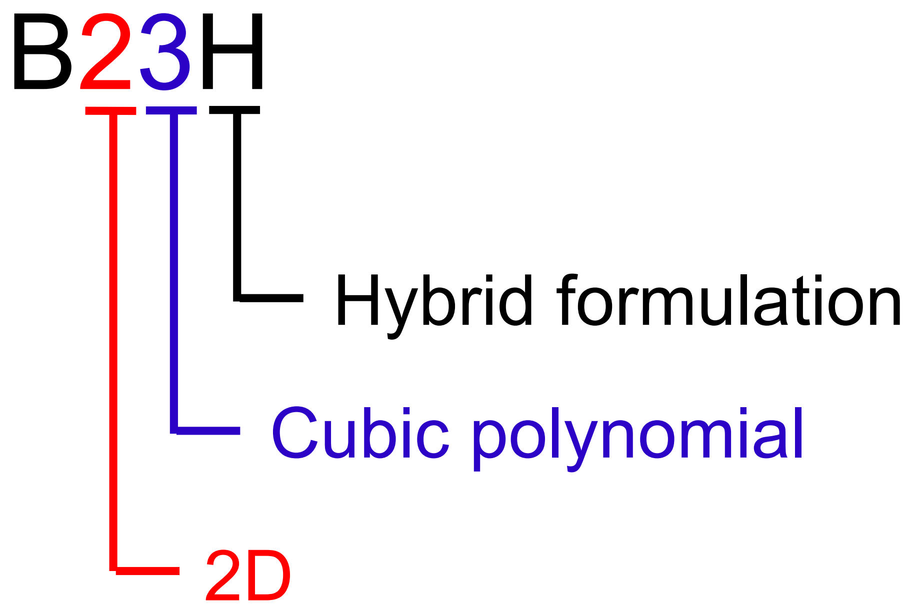
CLAMPED-CLAMPED BEAM

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



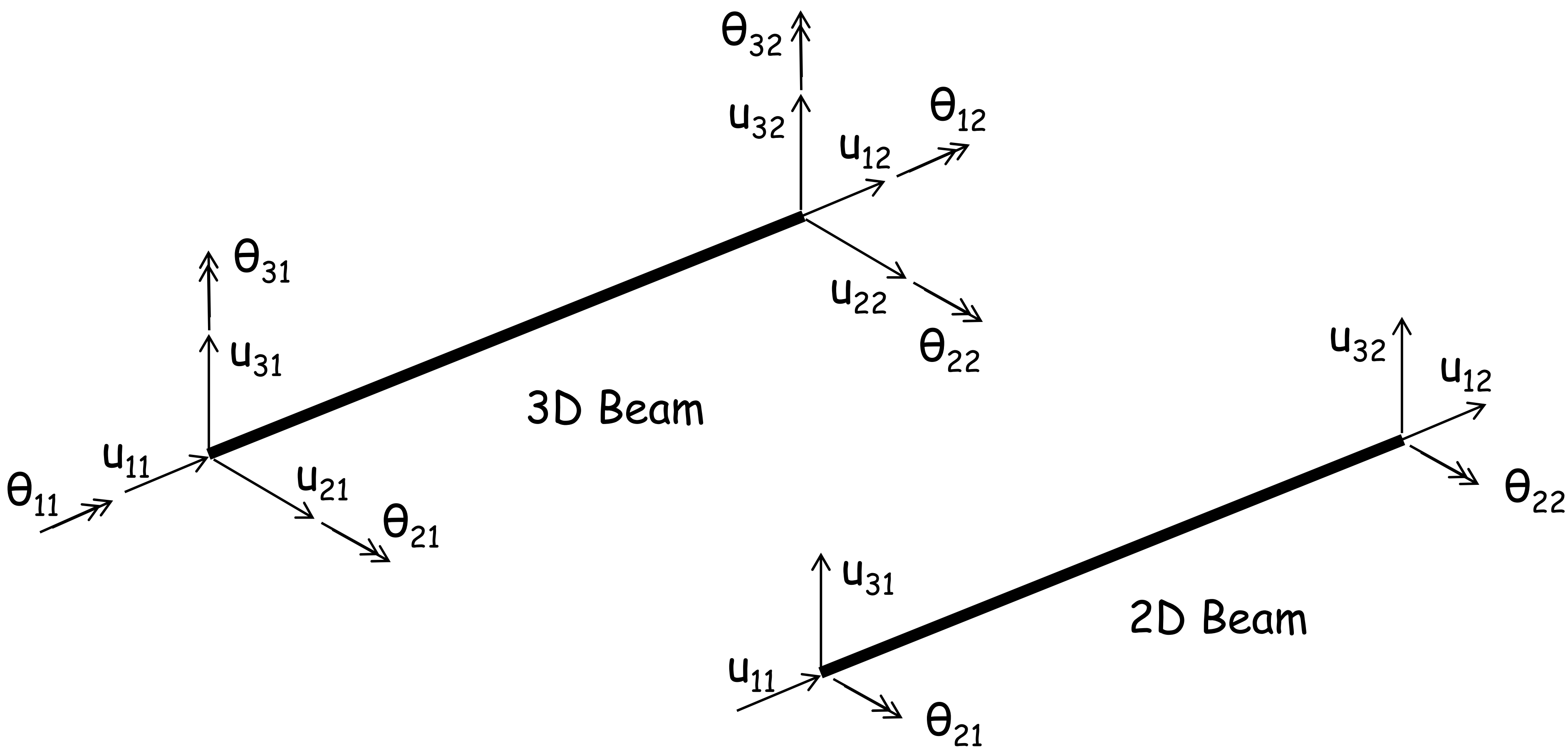
ABAQUS BEAM ELEMENT

- 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



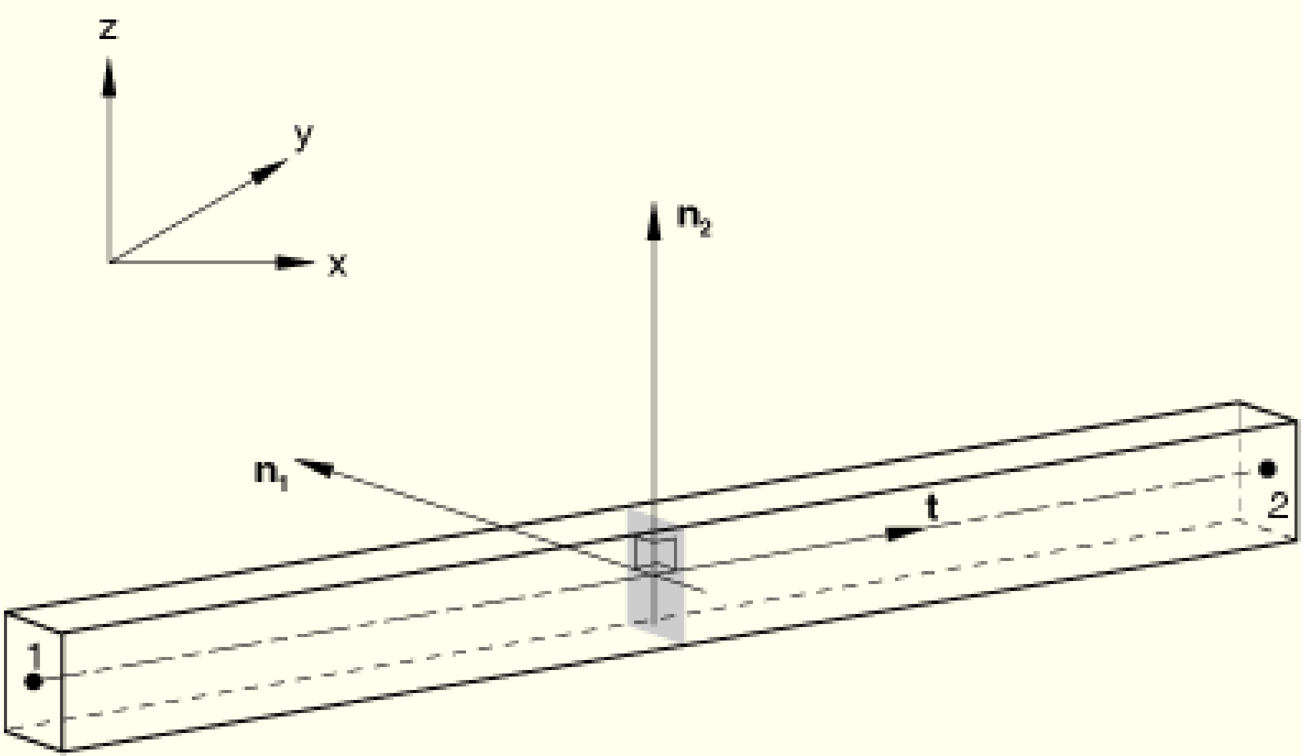
GEOMETRY MODELING

- 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)
- n_1 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

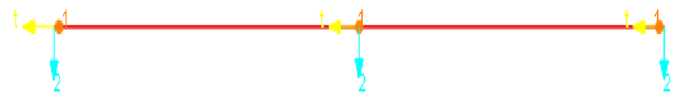


SECTION ORIENTATION cont.

CASE 1

Direction of wire geometry (t)

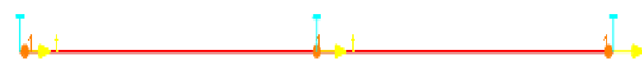
Assigned beam orientation



CASE 2

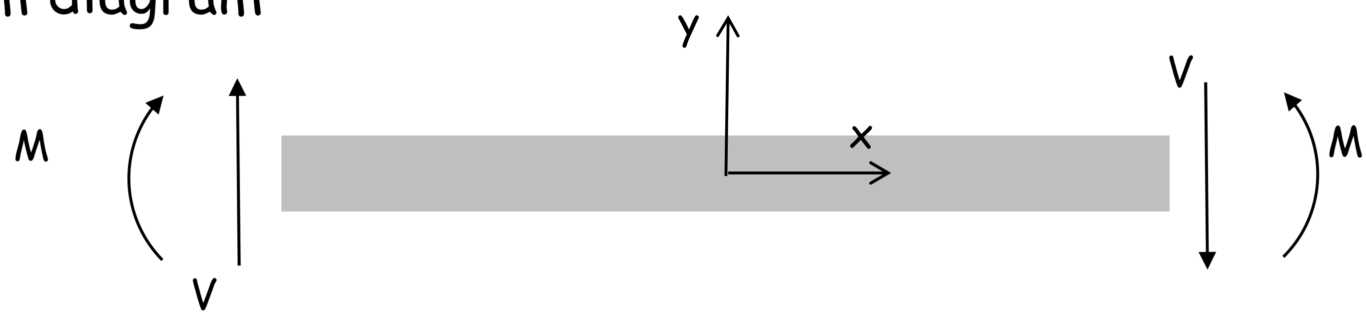
Direction of wire geometry (t)

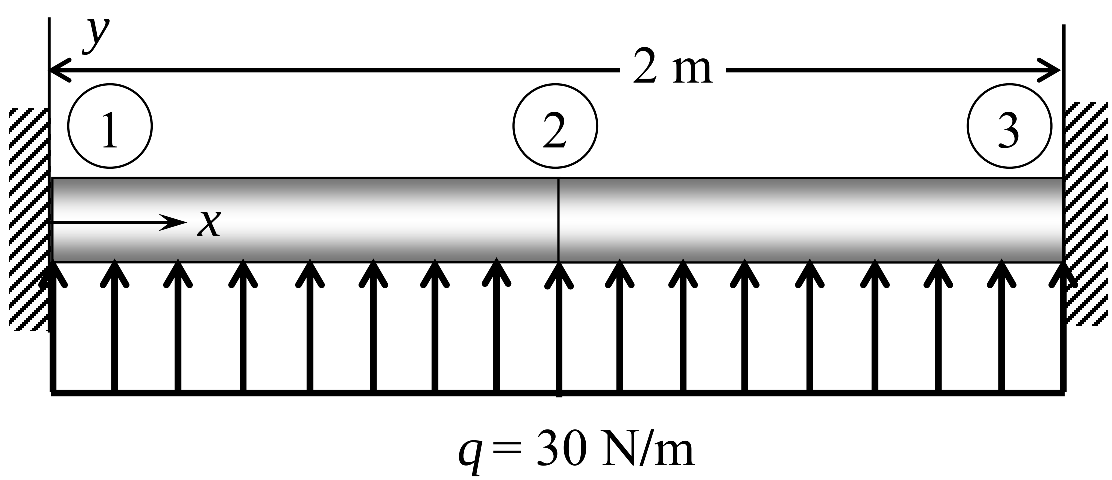
Assigned beam orientation



SIGN CONVENTION

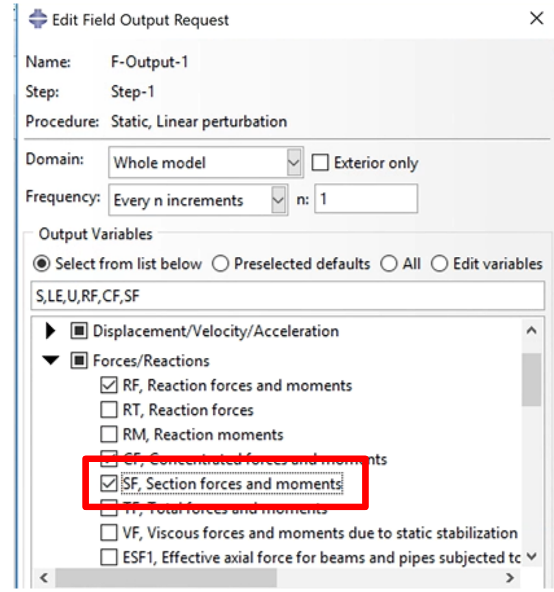
- Sign Convention for force and moment
  - Must not be confused with sign convention for displacements
  - Users must make sure which sign convention is used in a FEA software
  - Shear force and bending moment sign convention of ABAQUS for beam diagram





SIMULATION SETUP

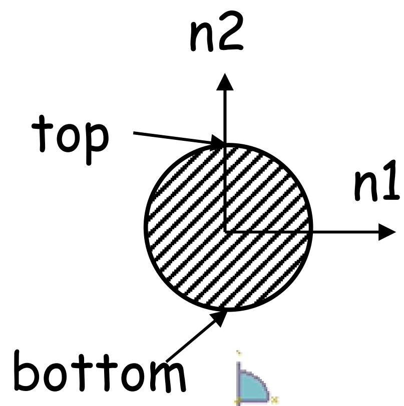
- Assembly, Instance
- Steps
  - Linear perturbation, Static
- Field output request
  - Check SF, Section forces and moments

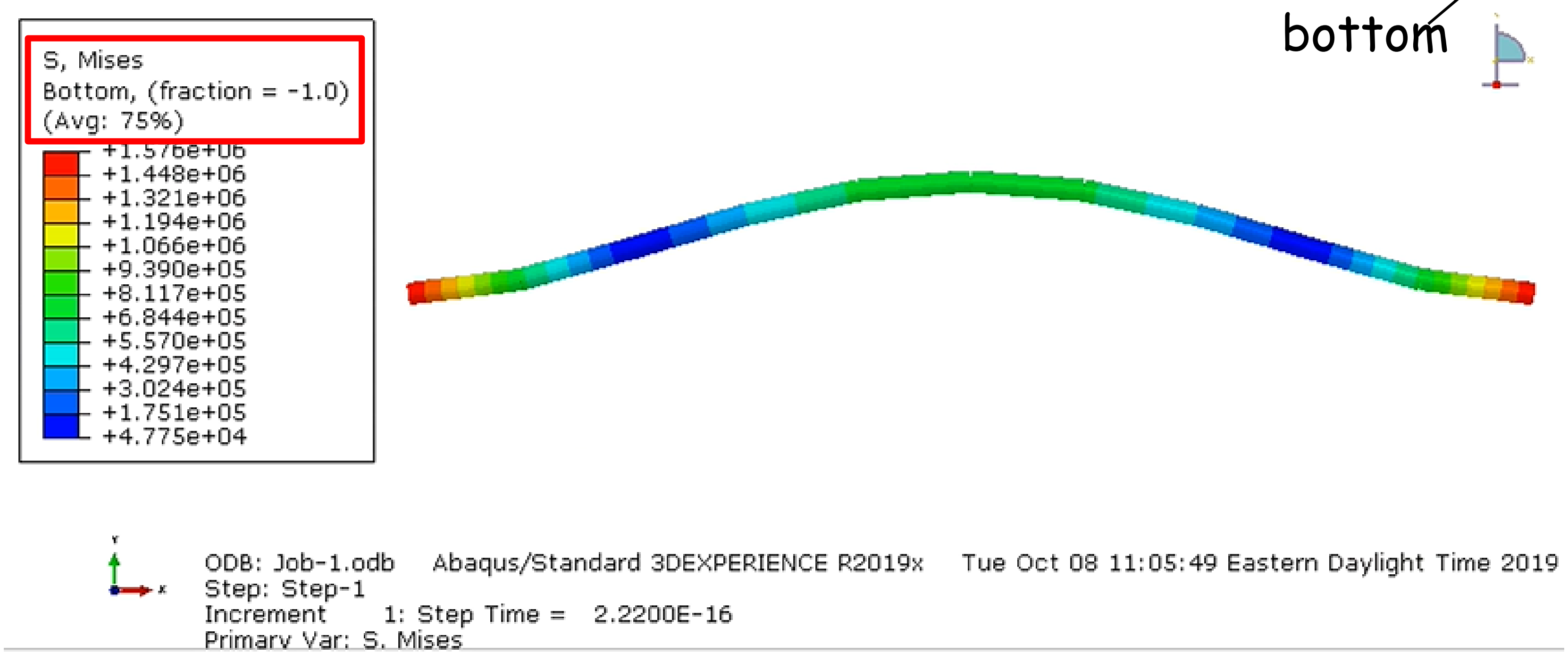


- BCs
  - Initial, Encastre (clamped)
- Loads
  - Mechanical, Pressure (force/length for beam) or Line load (force/length), select upward, Uniform, 30
- Mesh
  - Element type, "Classical beam theory" or Euler beam theory, Cubic polynomial, (B23), Global element size = 0.2.
  - Generate elements: Mesh/Parts

POSTPROCESSING

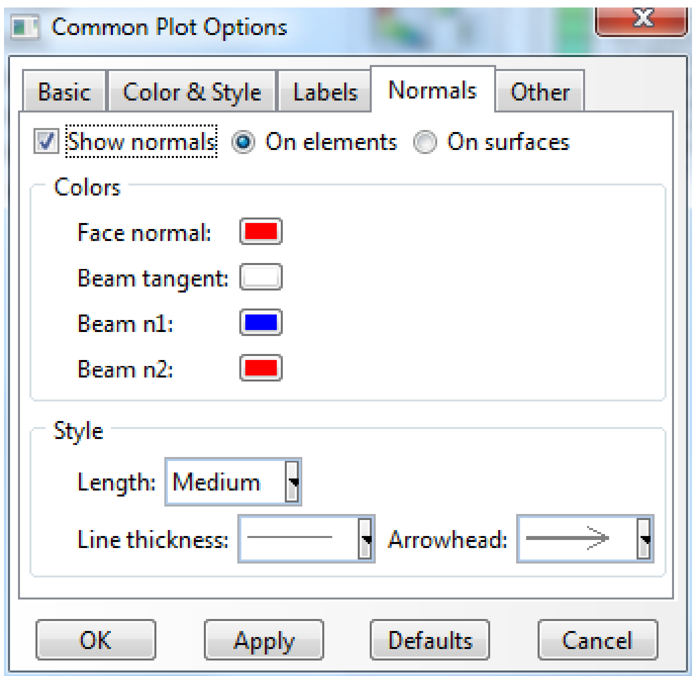
- Analysis, Create Job, Data Check, Submit
- Results
- Deformed plot, Stress plots
  - Result/Field output, Section points, Top or Bottom
  - View/ODB Displace Option/Render beam profiles

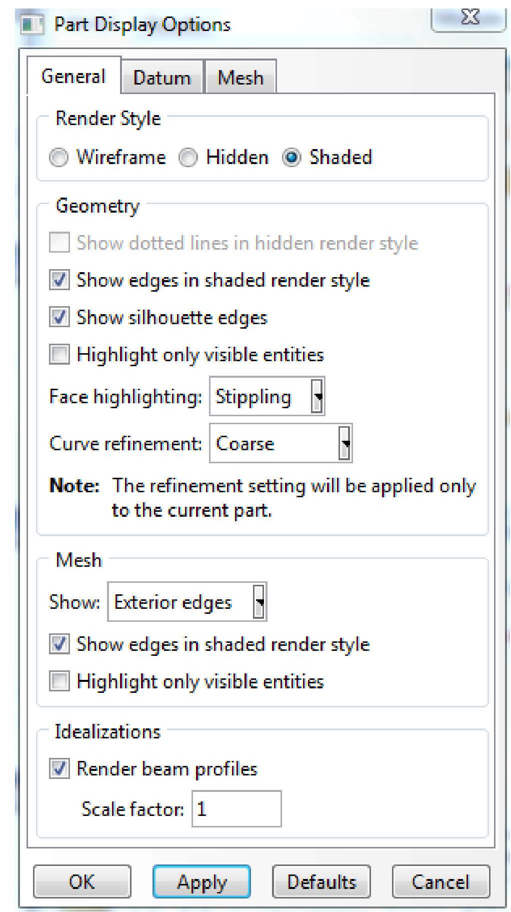




SHOWING ELEMENT NORMAL

- Common Plot Options
  - Normals, check “Show normals”, “On element”

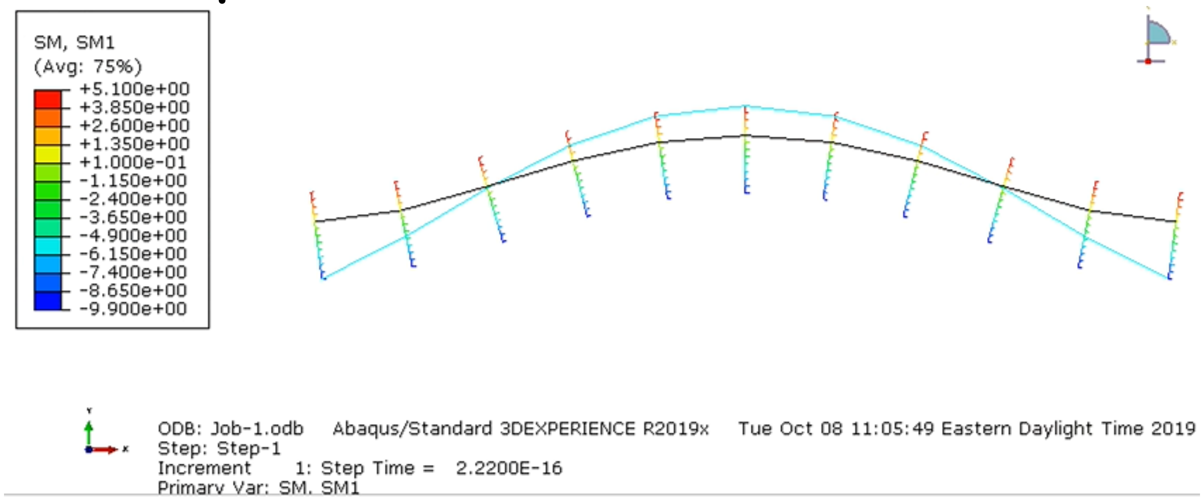




- Module part, View, Part Display Options
  - check “Render Beam Profile”

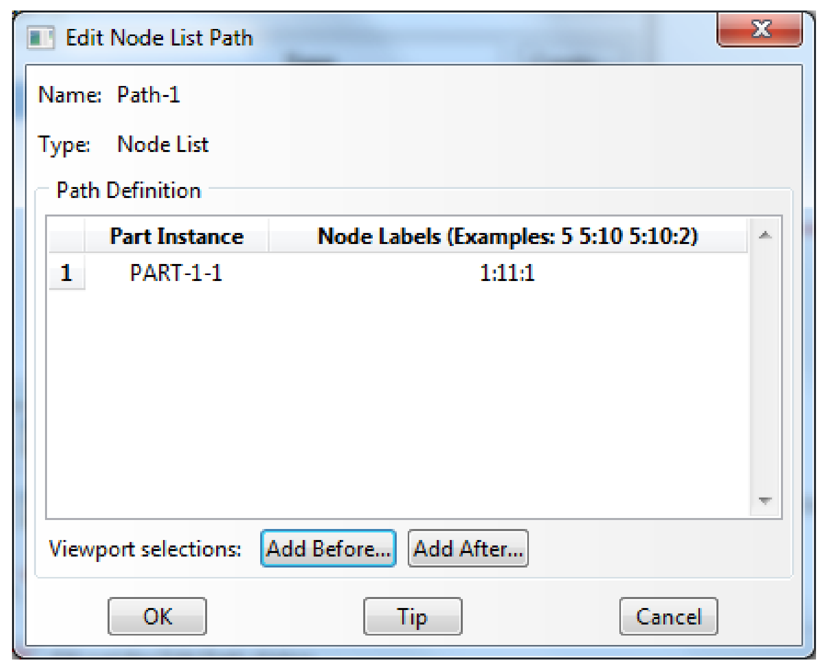
BENDING MOMENT DIAGRAM

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



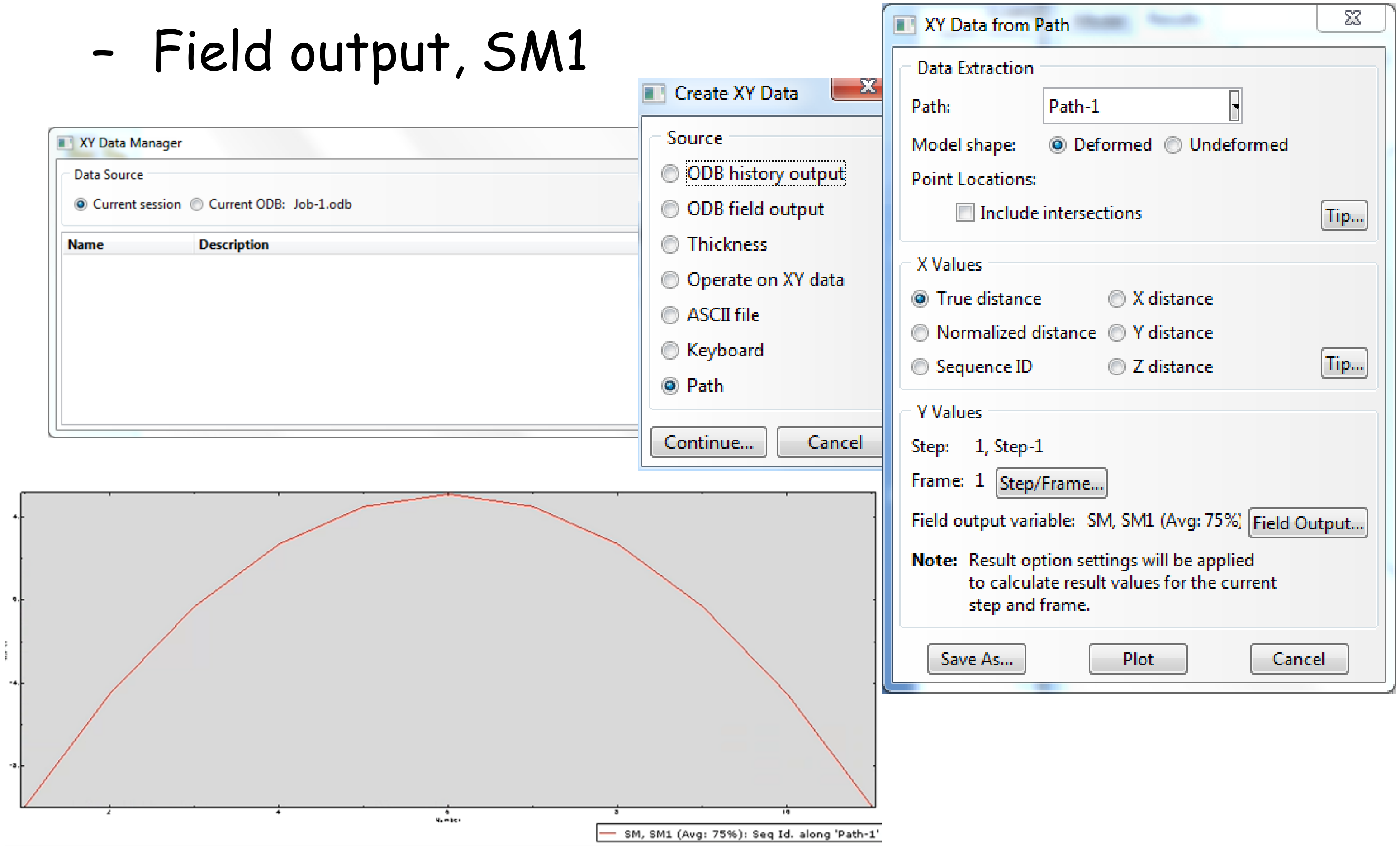
Bending moment diagram

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



BENDING MOMENT DIAGRAM cont.

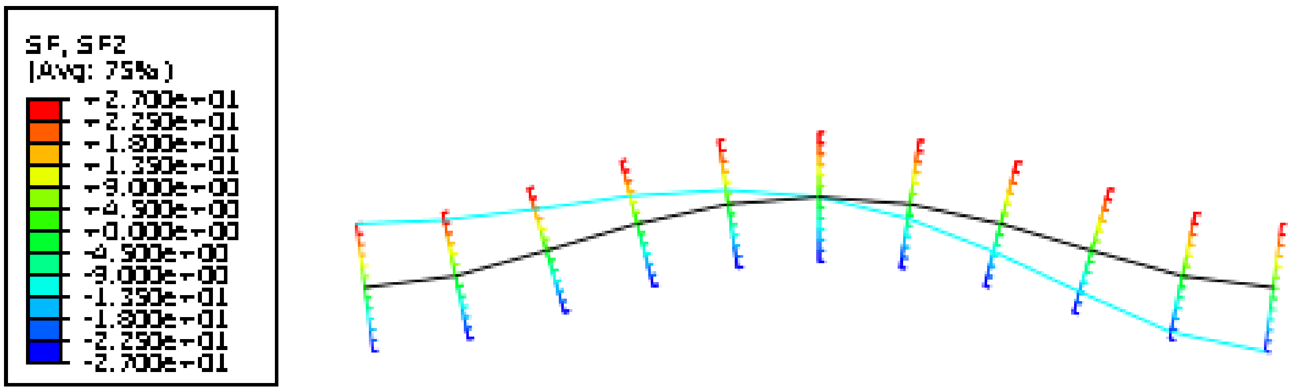
- XYData, path, X Values = Sequence ID
  - Field output, SM1



- 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

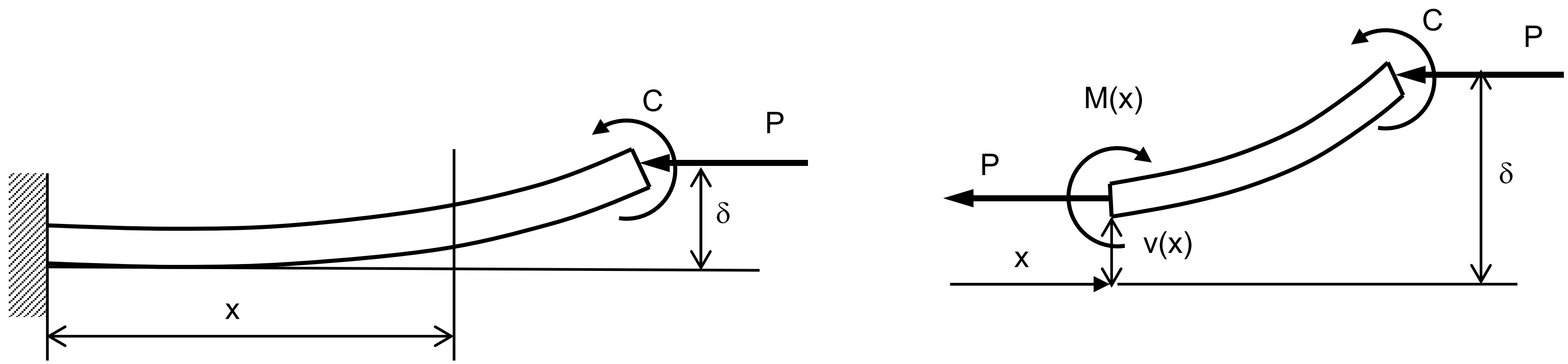
- Change element type
  - Element type, "Timoshenko beam theory" or Mindlin beam theory, Shear flexible, (B21), Global element size = 0.2
- Field Output
  - Shear force output (SF2) is available



Buckling Tutorial

Buckling of Column

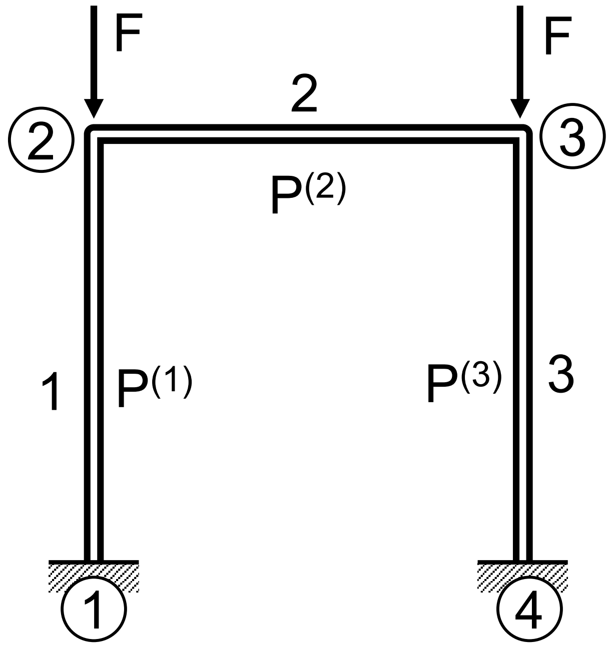
- In plane beam, we assume that axial deformation (bar) is independent of bending deformation (beam)
- This is normally true, but when the deformation is large or the force is large, the coupling effect become important



- 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?

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

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

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

$$[\mathbf{k}_{inc}^{(e)}] = \frac{P^{(e)}}{30L}\begin{bmatrix} 36 & 3L & -36 & 3L \\ 3L & 4L^2 & -3L & -L^2 \\ -36 & -3L & 36 & -3L \\ 3L & -L^2 & -3L & 4L^2 \end{bmatrix} \begin{matrix} v_i \\ \theta_i \\ v_j \\ \theta_j \end{matrix}$$

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

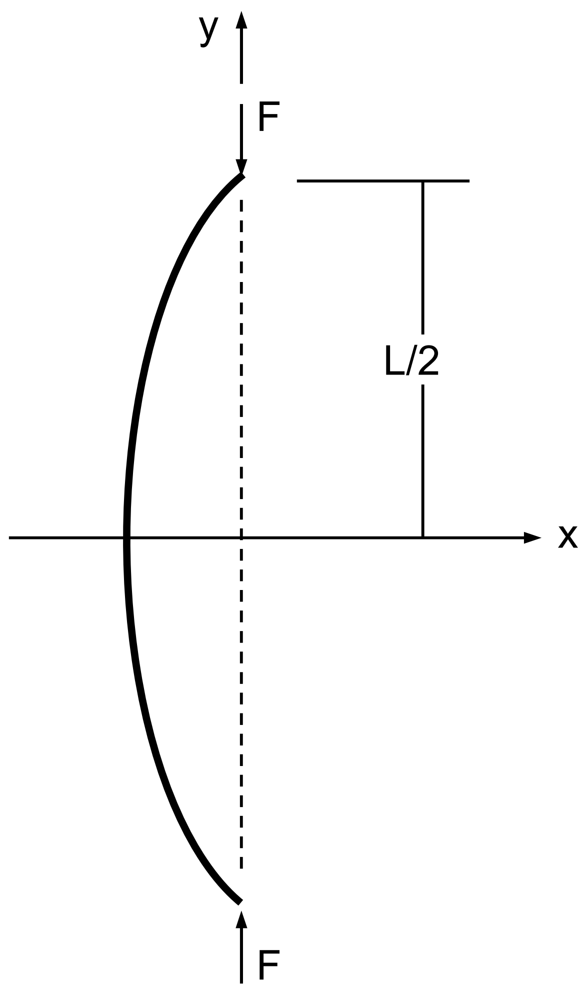
$$P_{cr} = \lambda \cdot F$$

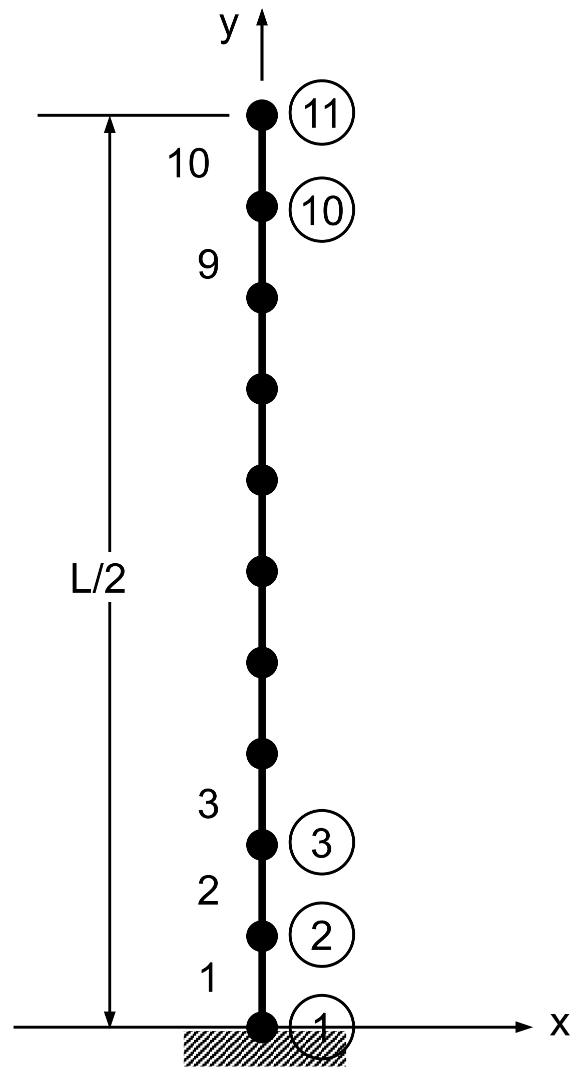
Buckling with Hinged Ends

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

$$P_{cr} = \frac{\pi^2 EI}{L^2}$$

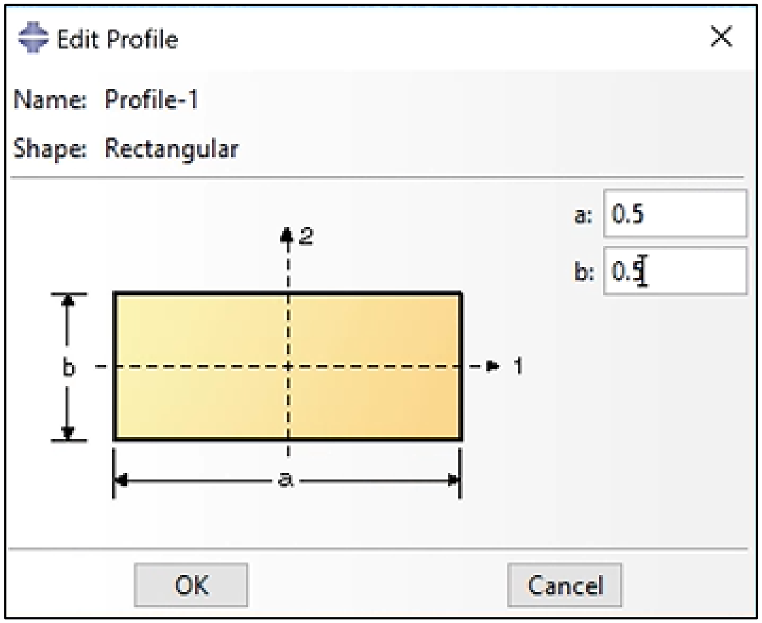
- Use symmetry (half model)





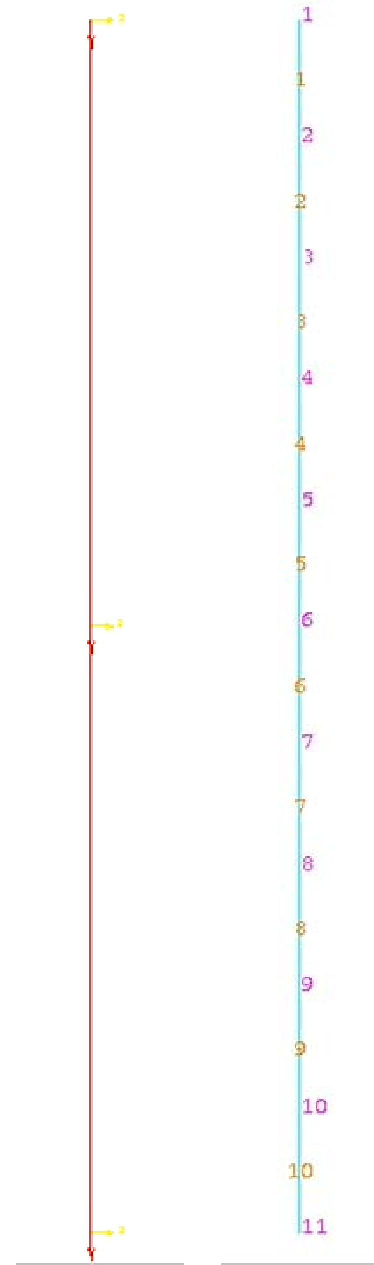
GEOMETRY MODELING

- 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



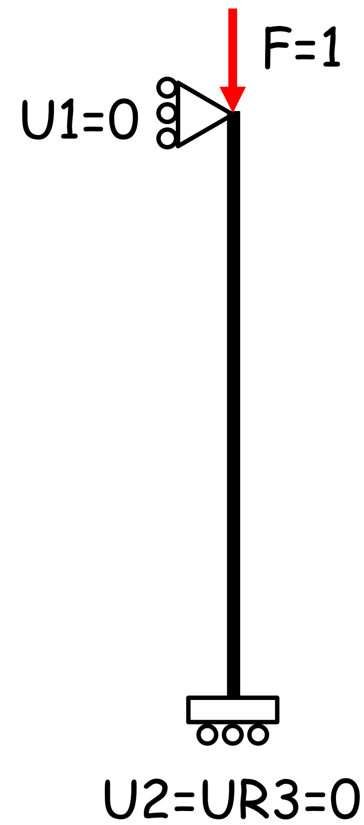
MESHING

- 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



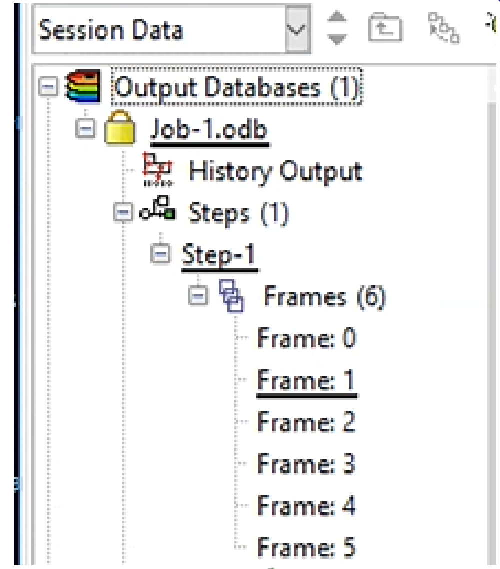
MESHING

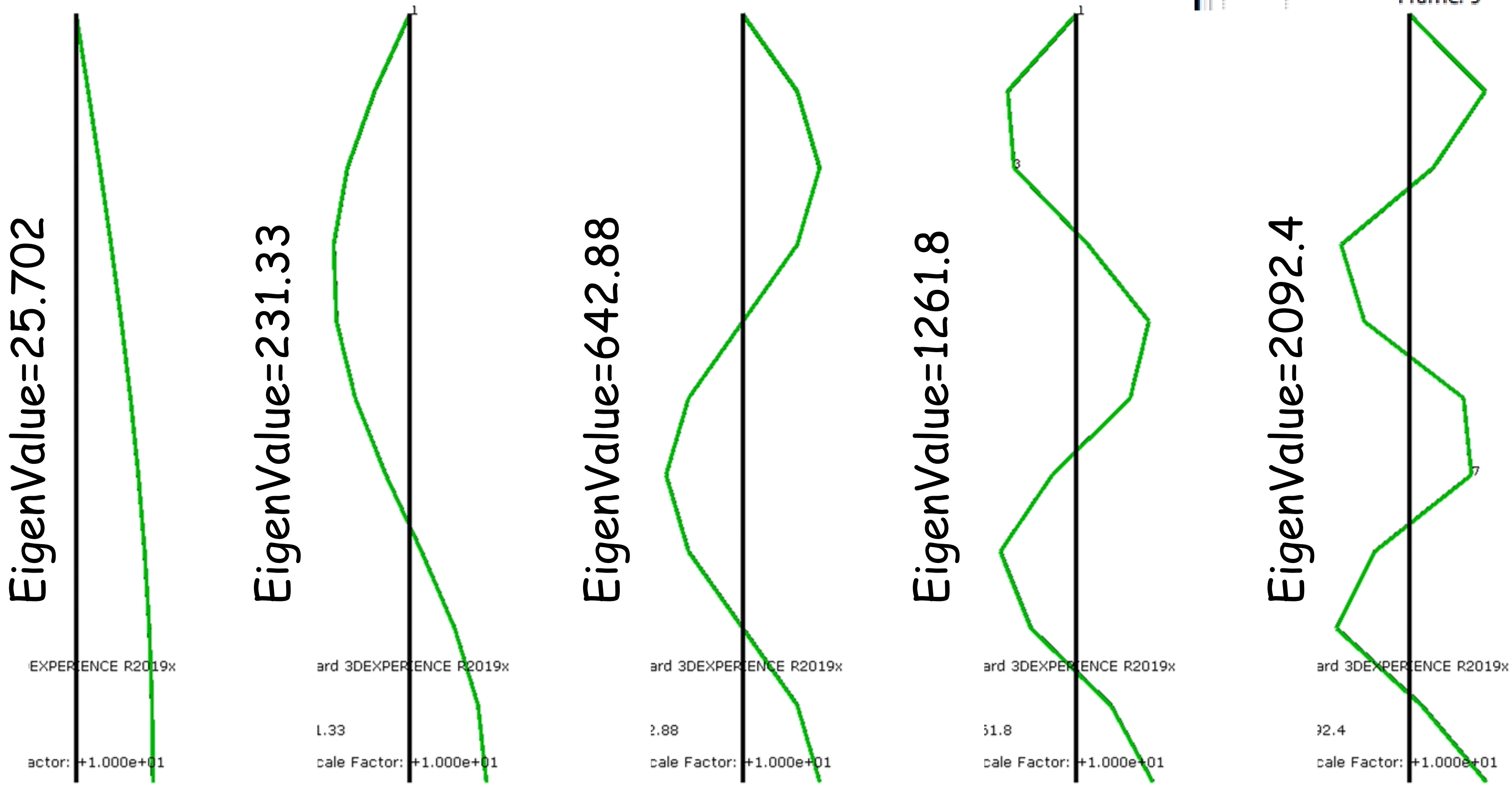
- Assembly, Instance, Part-1, OK
- Steps, Linear perturbation, Buckle, Number of eigenvalues requested = 5
- BCs, Step=Initial, Displacement/Rotation, "Select top node", Done, U1, OK
- BCs, Step=Initial, Displacement/Rotation, "Select bottom node", Done, U2, UR3, OK
- Loads, Step-1, Concentrated force,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



POSTPROCESSING

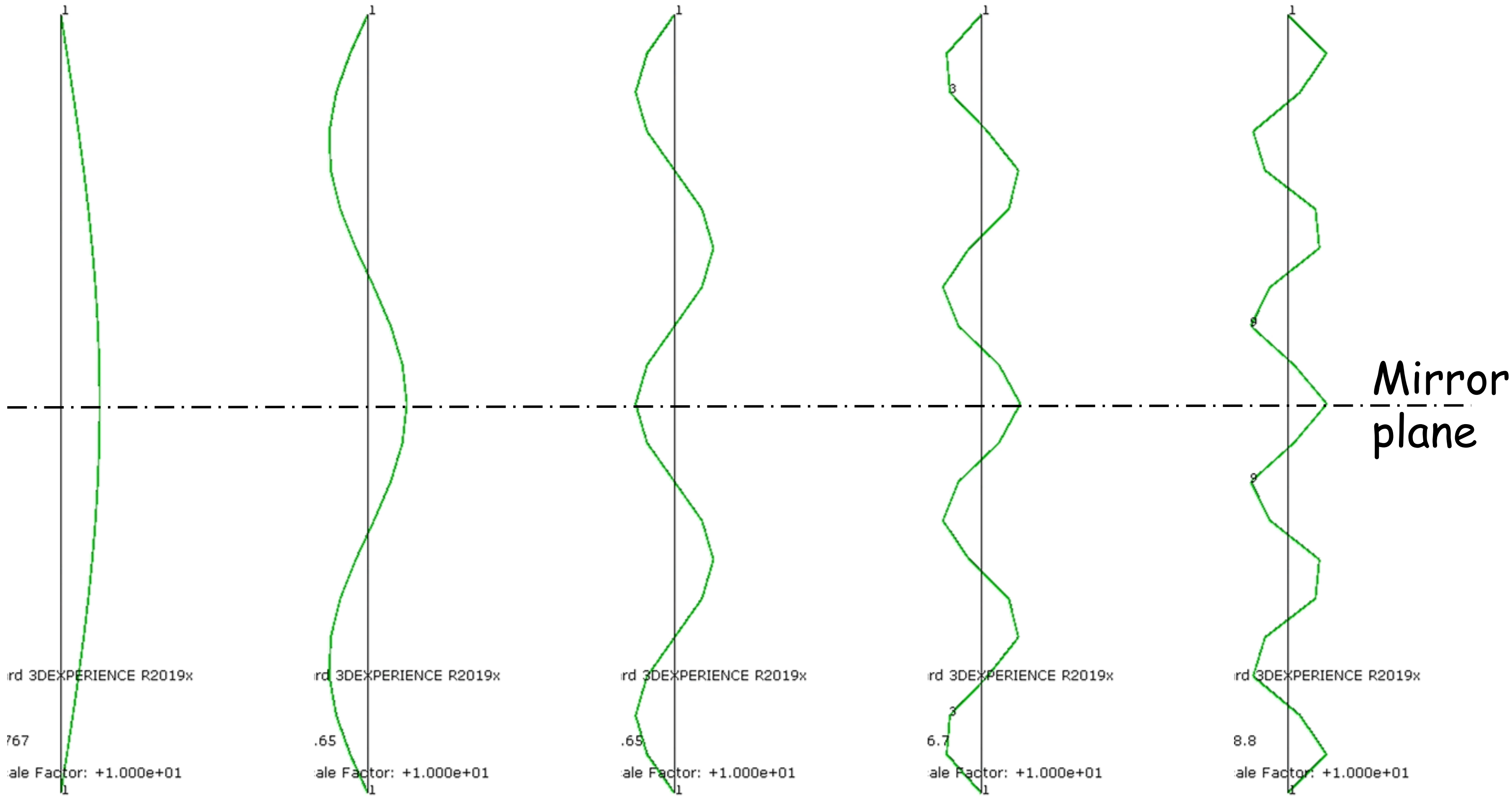
- Output Database, Job-1.odb, Steps, Step-1, Frames, “Select Frame:1”
- Plot Deformed Shape, “EigenValue = 25.702”
- Repeat for other frames





MIRROR IMAGE

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



UNDERSTAND BUCKLING LOAD

Eigenvalue		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!!