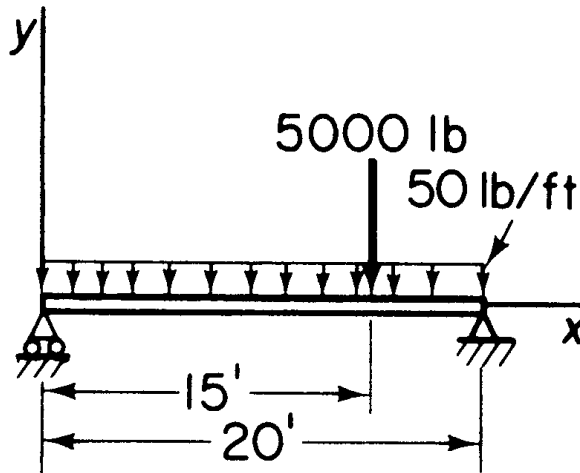


ABAQUS Tutorial - Beam Bending

Consider the beam bending problem:



Assume that the beam is made of steel ($E=30 \times 10^6$ psi, $G=11.5 \times 10^6$ psi) and has a 2" deep x 5" high rectangular cross section ($I_z=(2)(5^3)/12=20.83$ in⁴, $I_y=(5)(2^3)/12=3.333$ in⁴). Determine the maximum deflection and stress in the bar and the using 8 beam elements. Compare the solution to the beam theory solution.

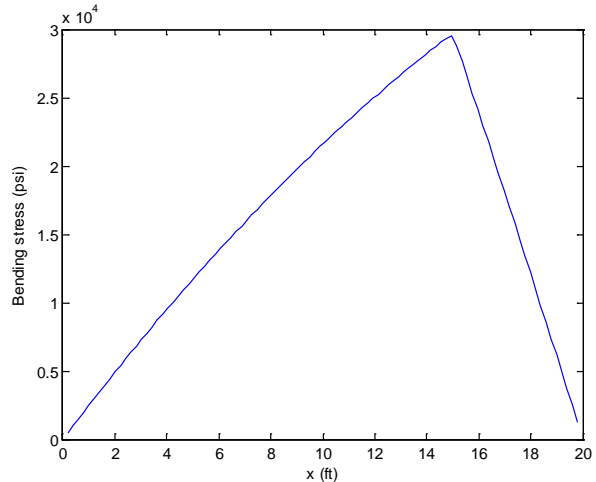
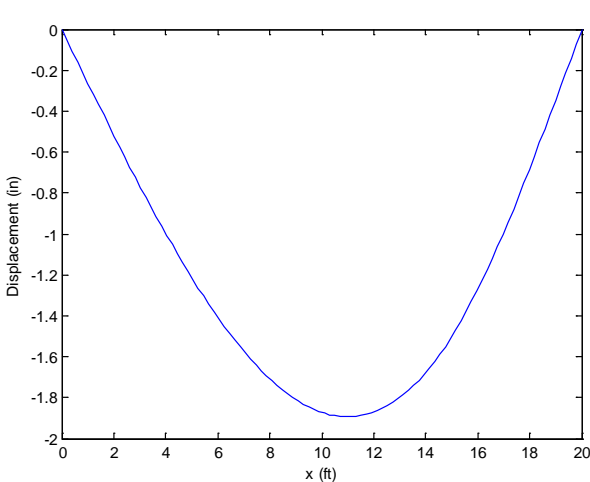
Beam theory solution

Beam theory gives the following displacement solution:

$$v(x) = \frac{Pbx}{6EIL} (x^2 + b^2 - L^2) + \frac{wx}{24EI} (2Lx^2 - x^3 - L^3), \quad 0 \leq x \leq a$$

$$v(x) = \frac{Pa(L-x)}{6EIL} (x^2 + a^2 - 2Lx) + \frac{wx}{24EI} (2Lx^2 - x^3 - L^3), \quad a \leq x \leq L$$

where $v(x)$ is the displacement, P is the concentrated force (-5000 lb), x is the distance from the left end of the beam, EI is the flexural stiffness of the beam, w_0 is the uniform distributed load (-50 lb/ft = -4.167 lb/in), $a=15$ ft and $b=5$ ft. The displacement field and bending stress distribution predicted by beam theory are shown below. Note that the maximum deflection, approximately -1.89 in, occurs between $x=11$ ft and $x=12$ ft and the maximum bending stress is approximately 29,700 psi at $x=15$ ft.



Finite Element solution (ABAQUS)

Start => Programs => ABAQUS 6.9-1 => ABAQUS CAE
 File => Set Work Directory => select folder for Abaqus generated files
 Select 'Create Model Database'
 File => Save As => save .cae file in Work Directory

Module: Sketch

Sketch => Create
 Add=> Point => enter coordinates (0,0), (180,0), (240,0), => select 'red X'
 Add => Line => Connected Line => select point at (0,0) with mouse, then (180,0), then (240,0),
 right click => Cancel Procedure => Done

Module: Part

Part => Create => select 2D Planar, Deformable, Wire, Approx size 200 => Continue
 Add => Sketch => select 'Sketch-1' => Done => Done

Module: Property

Material => Create => Name: Material-1, Mechanical, Elasticity, Elastic => set Young's modulus = 30e6, Poisson's ratio = 0.3 => OK
 Profile => Create => Generalized => A=10, $I_1 = 20.83$, $I_{12}=0$, $I_2=3.333$, J=0 => OK
 Section => Create => Name: Section-1, Beam, Beam => Continue => Section Integration – Before Analysis => Profile Name: Profile-1 => Linear Properties => E=30e6, G=11.54e6 => Output Points => enter $(x_1, x_2) = (0,-2.5)$ and $(x_1, x_2) = (0,2.5)$ => 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 => select “Dependent (mesh on part)” => OK

Module: Step

Step => Create => Name: Step-1, Initial, Static, General => Continue => nlgeom off => OK

Module: Load

Load => Create => Name: Step-1, Step: Step 1, Mechanical, Line Load => Continue => select full model => Done => set Component 1 =0, Component 2 = -4.167 => OK

Load => Create => Name: Step-1, Step: Step 1, Mechanical, Concentrated Force => Continue => select point at (180,0) => Done => set CF2=-5000 => OK

BC => Create => Name: BC-1, Step: Step-1, Mechanical, Displacement/Rotation => Continue => select point at (0,0) => Done => U2=0 => OK

BC => Create => Name: BC-1, Step: Step-1, Mechanical, Displacement/Rotation => Continue => select point at (240,0) => Done => U1=U2=0 => OK

Module: Mesh

In model tree, expand Model-1 => Parts => Part-1 => double-click on Mesh

Seed => Edge by Size => select full model by dragging mouse => Done => Element Size=30 => 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

Plot => Deformed Shape

Common Options => Labels => select 'Show element labels: Black' and 'Show node labels: Black'

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 => Top and Bottom

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

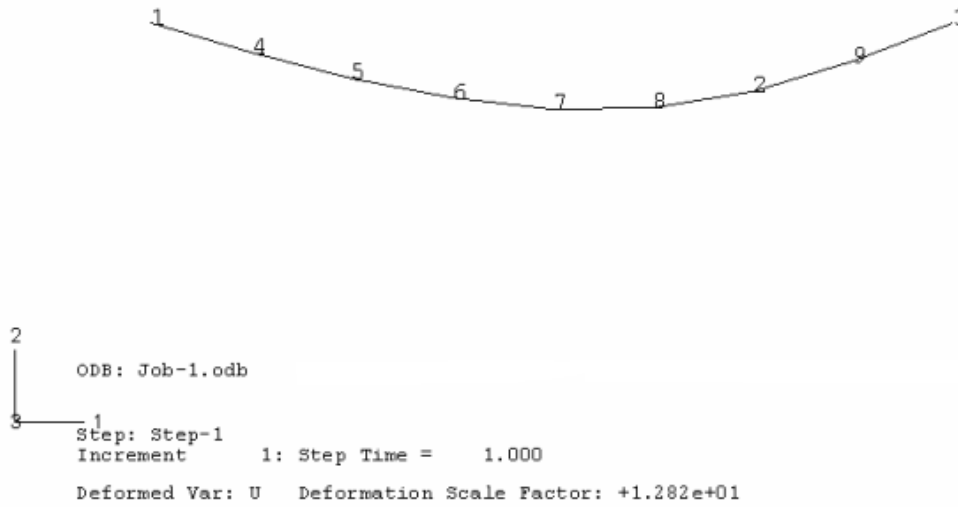
Report => Field Output => Setup => Number of Significant Digits => 6

Report => Field Output => Variable => Position: Unique Nodal => select U: Spatial Displacements, UR3: Rotational Displacements, S: Max. Principal => Apply

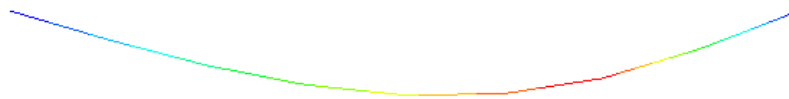
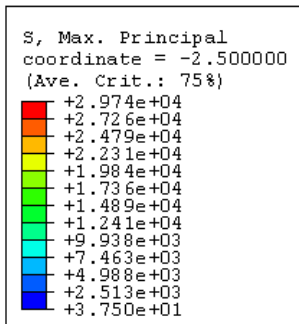
Cut and paste tabulated results from 'abaqus.rpt' file to MS Word document.

Results:

Deformed Mesh



Bending Stress Contours



Tabulated Output:

Node Label	U.Magnitude @Loc 1	U.U1 @Loc 1	U.U2 @Loc 1	UR3 @Loc 1	S.Max. Prin @Loc 5	S.Max. Prin @Loc 6
1	1.68753E-33	0.	-1.68753E-33	-21.8403E-03	-37.5029	37.5029
2	1.50122	0.	-1.50122	17.0402E-03	-29.7377E+03	29.7377E+03
3	4.18754E-33	0.	-4.18754E-33	29.0403E-03	-37.5025	37.5025
4	642.834E-03	0.	-642.834E-03	-20.6103E-03	-6.11263E+03	6.11263E+03
5	1.21322	0.	-1.21322	-17.0402E-03	-11.7377E+03	11.7377E+03
6	1.64365	0.	-1.64365	-11.3101E-03	-16.9128E+03	16.9128E+03
7	1.87202	0.	-1.87202	-3.6E-03	-21.6378E+03	21.6378E+03
8	1.84165	0.	-1.84165	5.91011E-03	-25.9128E+03	25.9128E+03
9	840.834E-03	0.	-840.834E-03	26.0103E-03	-15.1126E+03	15.1126E+03
Minimum At Node	1.68753E-33 1	0. 9	-1.87202 7	-21.8403E-03 1	-29.7377E+03 2	37.5025 3
Maximum At Node	1.87202 7	0. 9	-1.68753E-33 1	29.0403E-03 3	-37.5025 3	29.7377E+03 2