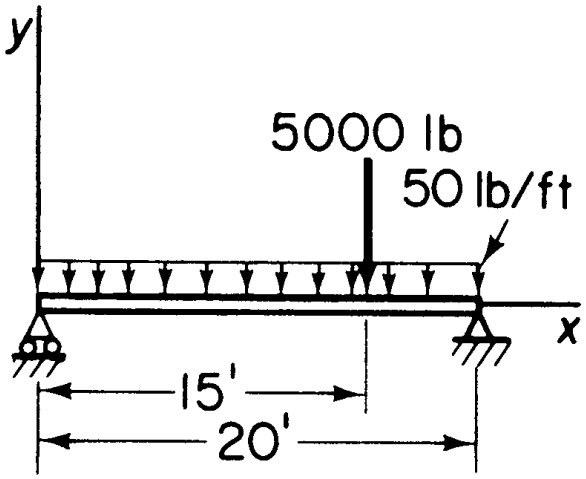
Tutorial 2. Beam Bending Analysis

Consider the beam bending problem:



Assume that the beam is made of steel (E=30x10^6 psi, G=11.5x10^6 psi) and has a 2" deep x 5" high rectangular cross section (I_z=(2)(5^3)/12=20.83 in^4, I_y=(5)(2^3)/12=3.333 in^4). Determine the maximum deflection and stress in the bar and the using 8 beam elements. Compare the solution to the beam theory solution.

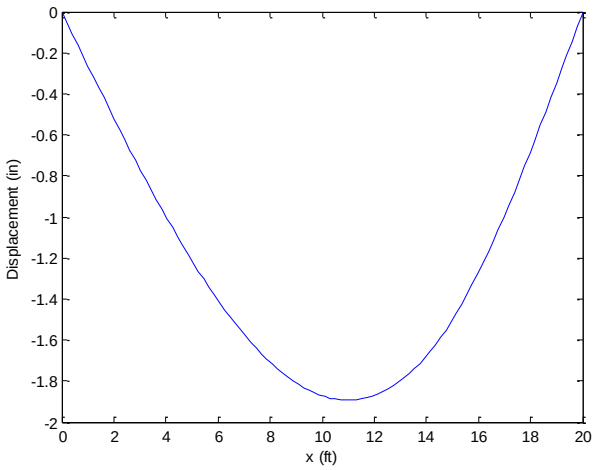
Beam theory solution

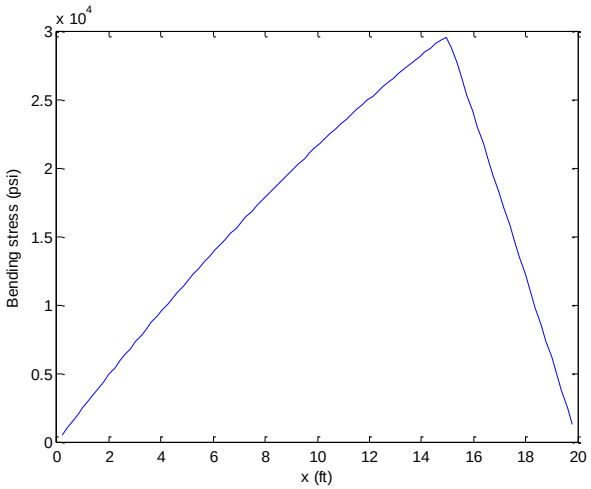
Beam theory gives the following displacement solution:

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

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

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_o 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

Start => All Programs => Dassault Systems SIMULIA Abaqus => Abaqus CAE => Create Model Database With Standard/Explicit Model
File => Set Working Directory => Browse to find desired directory => OK
File => Save As => save beam_tutorial.cae file in Work Directory

Module: Sketch
Sketch => Create => continue
Add => Line => Connected Line => enter coordinates (0,0), (180,0), (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
Assign Section => select all elements by dragging mouse => Done => Section-1 => Done => OK
Assign Beam Section Orientation => select full model => Done => n_1 direction = 0.0,0.0,-1.0 => OK =>Done

Module: Assembly
Instance => Create => Create instances from: Parts => Part-1 => Dependent (mesh on part) => OK

Module: Step

Step => Create => Name: Step-1, Initial, Static, General => Continue => accept default settings => OK

Module: Load

Load => Create => Name: Load-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

Set Model: Model-1, Object => Part: Part-1

Seed => Edges => select entire beam by dragging mouse => Done => Method: By size, Bias: None, Sizing Controls, Element Size=30 => OK => Done

Mesh => Element Type => select entire truss by dragging mouse => Done => Element Library: Standard, Geometric Order: Linear: Family: Beam, Cubic interpolation (B23)=> OK => Done

Mesh => Part => OK to mesh the part Instance: Yes

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

Job => Manager => Results (transfers to Visualization Module)

Module: Visualization

Viewport => Viewport Annotation Options => Legend => Text => Set Font => Size=14, Apply to: Legend, Title Block and State Block => OK => OK

View => Graphics Options => Viewport Background = Solid=> Color => White (click on black tile to change background color)

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

Plot => Deformed Shape

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

Result => Field Output => select S, Component: S11 => Section Points => Top and Bottom => OK

Plot=> Contours => On Deformed Shape

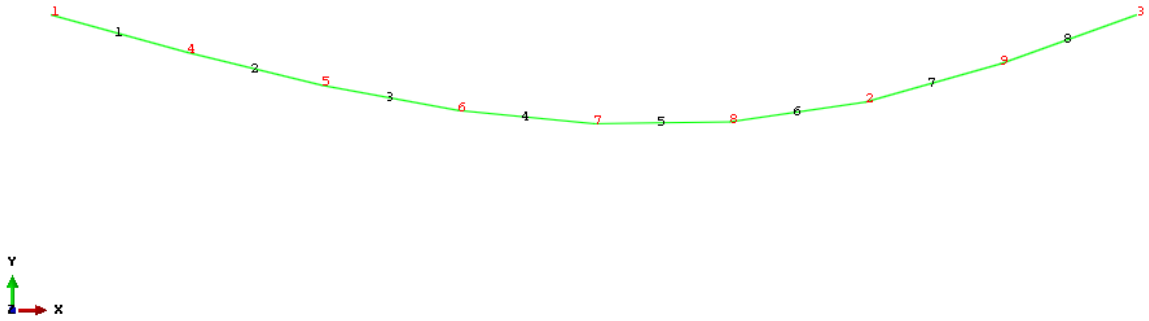
Report => Field Output => Variable => Position: Unique Nodal => select Spatial displacement: U2: Spatial Displacements, Rotational displacement: UR3 => OK

Report => Field Output => Variable => Position: Unique Nodal => select Stress components: S11, Section points - All => OK

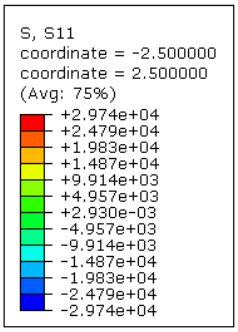
Cut and paste tabulated results from 'Abaqus.rpt' file to MS Word document.

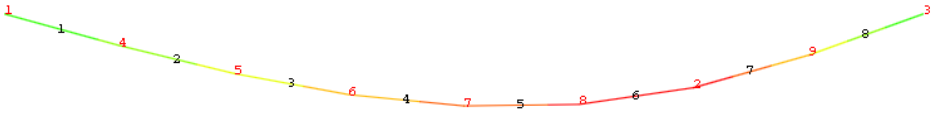
Results:

Deformed Mesh



Bending Stress Contours





Tabulated Output:

Node Label	S.S11 @Loc 1
1	37.5090
2	29.7425E+03
3	37.5085
4	6.11361E+03
5	11.7396E+03
6	16.9155E+03
7	21.6413E+03
8	25.9169E+03
9	15.1150E+03
Minimum	37.5085
At Node	3
Maximum	29.7425E+03
At Node	2
Total	127.259E+03

Node Label	U.U2 @Loc 1
1	-1.68753E-33
2	-1.50146
3	-4.18754E-33
4	-642.937E-03
5	-1.21341
6	-1.64391
7	-1.87232
8	-1.84194
9	-840.968E-03
Minimum	-1.87232
At Node	7
Maximum	-1.68753E-33
At Node	1
Total	-9.55695

Node Label	S.S11 @Loc 2
1	-37.5090
2	-29.7425E+03
3	-37.5085
4	-6.11361E+03
5	-11.7396E+03
6	-16.9155E+03
7	-21.6413E+03
8	-25.9169E+03
9	-15.1150E+03
Minimum	-29.7425E+03
At Node	2
Maximum	-37.5085
At Node	3
Total	-127.259E+03

Node Label	UR3 @Loc 1
1	-21.8438E-03
2	17.0429E-03
3	29.0450E-03
4	-20.6136E-03
5	-17.0429E-03
6	-11.3119E-03
7	-3.60058E-03
8	5.91106E-03
9	26.0144E-03
Minimum	-21.8438E-03
At Node	1
Maximum	29.0450E-03
At Node	3
Total	3.60058E-03