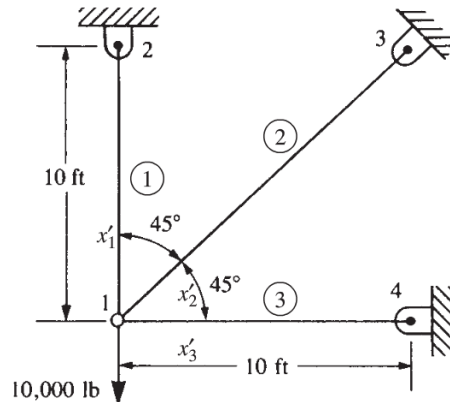# Tutorial 1. 2D Truss Analysis

Problem: Determine the nodal displacements and element stresses for the truss shown below
(ref. "A First Course in the Finite Element Method, 5th edition, Daryl L. Logan, 2012,
example 3.5, pp. 92-95). Use E=30x10^6 psi and A=2 in^2. Compare to text solution: u_1 =
0.414e-2 in, v_1 = -1.59e-2 in, element stresses = -1035, 1471, and 3965 psi.

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 truss_tutorial.cae file in Work Directory

Module: Sketch
Sketch => Create => Name: truss-demo => Continue
Add=> Point => enter coordinates (0,0), (120,0), (120,120), (0,120) => select 'red X'
View => Auto-Fit
Add => Line => Connected Line => select (0,0) node with mouse, then (120,0) node, right click
        => Cancel Procedure
Add => Line => Connected Line => select (0,0) node with mouse, then (120,120) node, right
        click => Cancel Procedure
Add => Line => Connected Line => select (0,0) node with mouse, then (0,120) node, right click
        => Cancel Procedure=> Done

Module: Part
Part => Create => select 2D Planar, Deformable, Wire => Continue
Add => Sketch => select 'truss_demo' => Done => Done

Module: Property
Material => Create => Name: Material-1, Mechanical, Elasticity, Elastic => set Young's
        modulus = 30e6, Poisson's ratio = 0.3 => OK
Section => Create => Name: Section-1, Beam, Truss => Continue => set Material: Material-1,
        Cross-sectional area: 2

Assign Section => select all elements by dragging mouse => Done => Section-1 => 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, Concentrated Force => Continue => select node at (0,0) => Done => set CF2: -10000 => OK
BC => Create => Name: BC-1, Step: Step-1, Mechanical, Displacement/Rotation => Continue => select nodes at (120,0), (120,120) and (0,120) using SHIFT key to select multiple nodes => Done => set U1: 0 and U2: 0

Module: Mesh
Set Model: Model-1, Object => Part: Part-1
Seed => Edges => select entire truss by dragging mouse => Done => Method: By number, Bias: None, Sizing Controls, Number of Elements: 1 => press Enter => Done
Mesh => Element Type => select entire truss by dragging mouse => Done => Element Library: Standard, Geometric Order: Linear: Family: Truss => 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 => 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 => Undeformed Shape
Plot => Deformed Shape
Plot => Contours => On Deformed Shape
Result => Options => unselect "Average element output at nodes"
Result => Field Output => Component: S11 => OK
Ctrl-C => Copies graphics window to clipboard => Paste in MS Word, etc.

Report => Field Output => Variable => Position: Unique Nodal => select U: Spatial
    Displacements => Apply => Unselect U
Report => Field Output => Variable => Position: Centroid => select S: Stress Components =>
    Click on '>' and unselect all stresses except S11 => Apply => Cancel
Open file 'Abaqus.rpt' and cut and paste desired results into MS Word
File => Save => enter desired file name (Abaqus will append .cae)
File => Exit

Results:
Deformed Mesh:
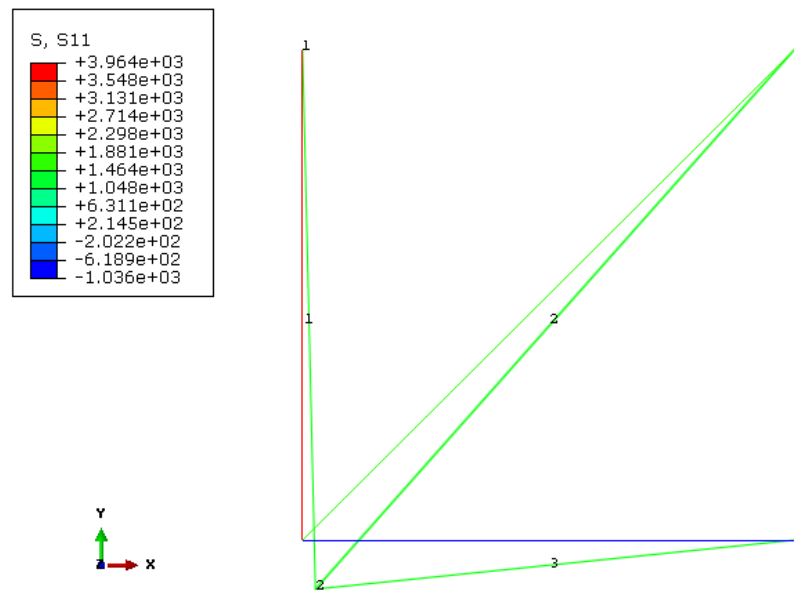
Tabulated Results (using cut and paste from Abaqus.rpt)

Node Label	U.Magnitude @Loc 1	U.U1 @Loc 1	U.U2 @Loc 1
1	0.	0.	-7.92893E-33
2	16.3899E-03	4.14214E-03	-15.8579E-03
3	0.	-2.07107E-33	-2.07107E-33
4	0.	2.07107E-33	0.

Element Label	S.S11 @Loc 1
1	3.96447E+03
2	1.46447E+03
3	-1.03553E+03