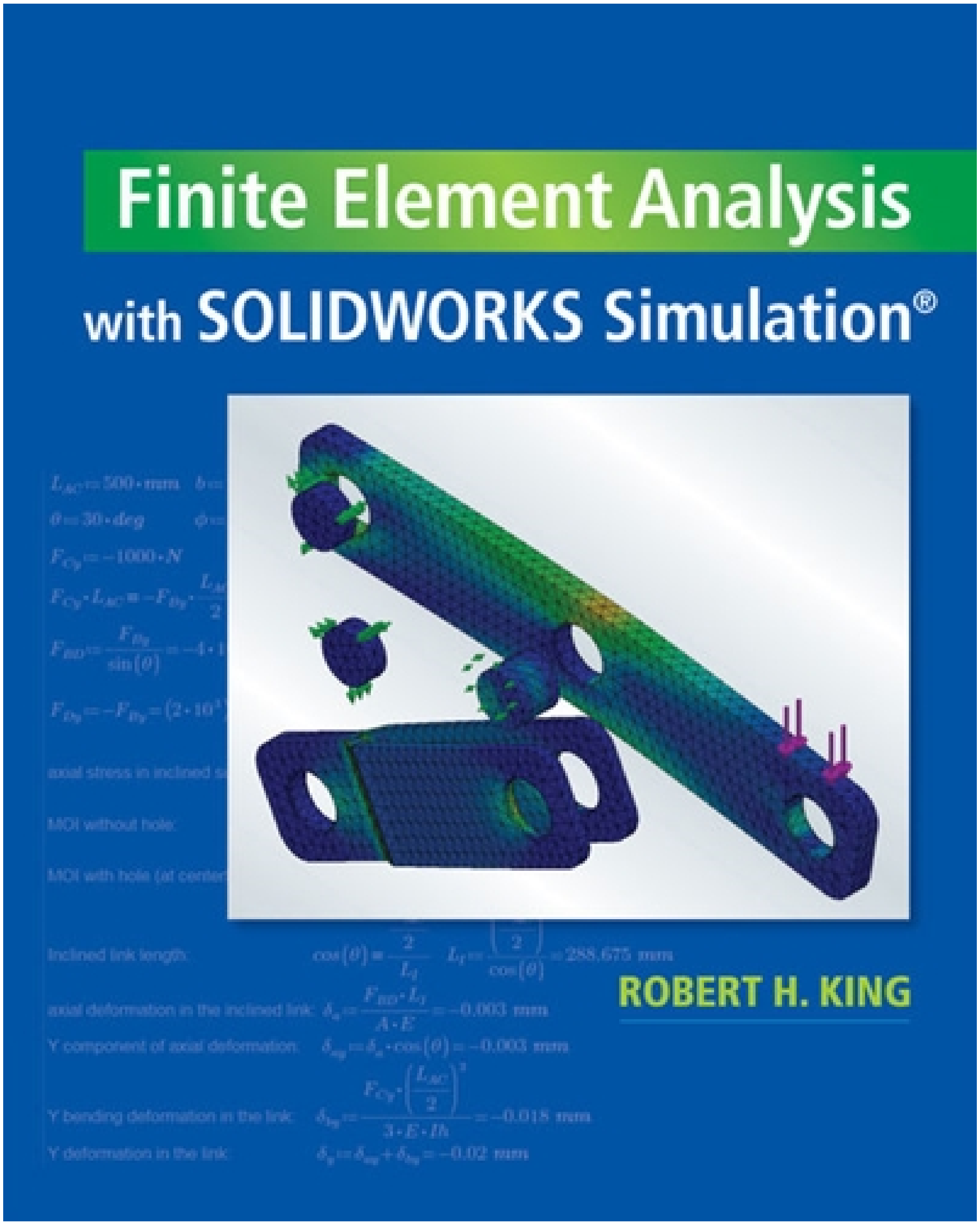
Solutions for Finite Element Analysis with SOLIDWORKS Simulation 1st Edition by King

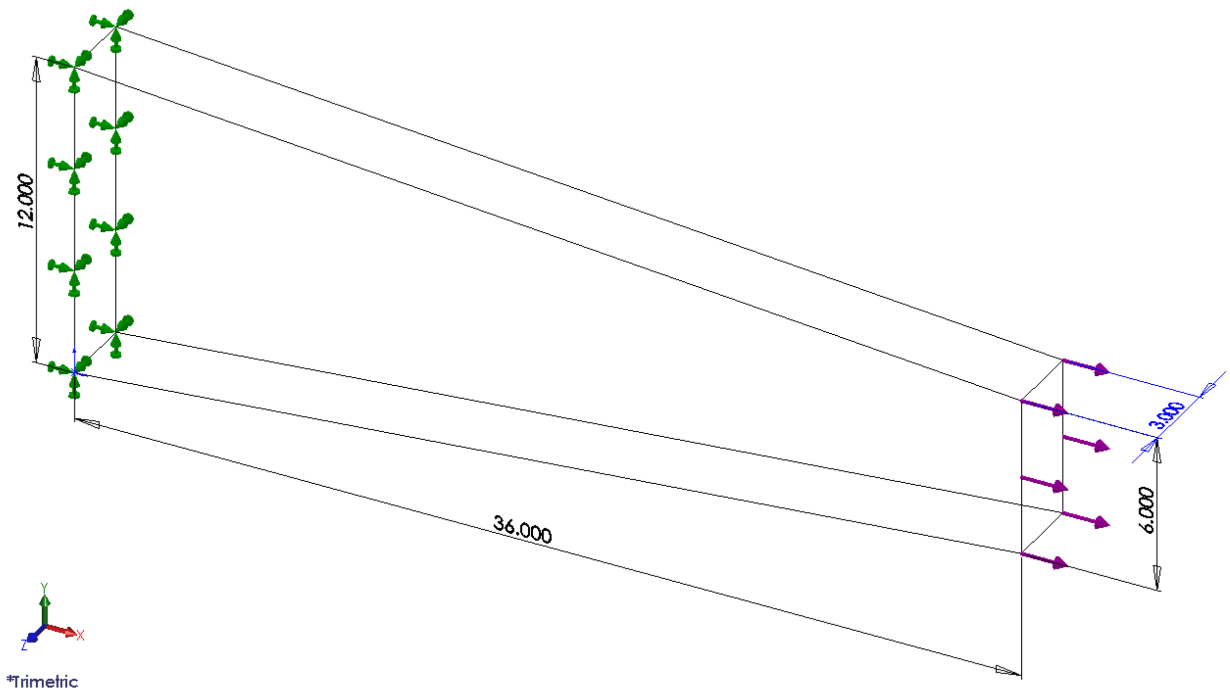
CLICK HERE TO ACCESS COMPLETE Solutions



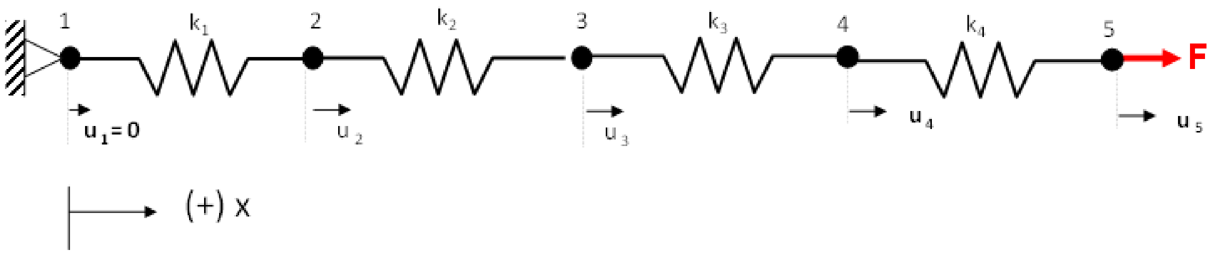
Solutions

Chapter 2 Exercises

2.1. Analyze the stress and displacement in the axially loaded tapered bar shown.



Dimension units are inches. The left-end face is fully fixed, and a tensile load of 180,000 lbf is applied uniformly on the right-end face. The bar is made of chrome stainless steel. Create an approximate model of the axially loaded tapered rod with four spring elements of equal length. The cross-section area should be constant throughout an element. Keep the z dimension constant for all four elements. The x dimension will be constant over an element, but each element will have a different y dimension. The y dimension is the average of the tapered rod y dimension over the length of each of the 1/4-length bar elements. Use the notation of u_1 for the displacement at node 1, which is the node at the restraint on the left end of the bar. Draw the 1D spring-element model. Number all of the nodes. Show the spring constants and their subscripts for each spring element. Show the force with an arrow attached to the appropriate node. Show the restraint attached to the appropriate node. Show the positive direction with a label and arrow. Show the location of the displacements and differentiate between them with subscripts.



2.2. Calculate the numerical values of all spring constants for the tapered-bar model described in Exercise 2.1.

	Element 1	Element 2	Element 3	Element 4
X dimension, L	9 in	9 in	9 in	9 in
Y dimension, h	11.25 in	9.75 in	8.25 in	6.75 in
Z dimension, b	3 in	3 in	3 in	3 in
Cross section area, A	33.75 in^2	29.25 in^2	24.75 in^2	20.25 in^2
Spring constant, K	8.158e7 lbf/in	7.017e7 lbf/in	5.983e7 lbf/in	4.895e7 lbf/in

units:Custon IPS
1/24/2012

$ORIGIN := 1$

Worksheet>Options>Calculaltion Set strict singularity checking for matrices

GIven

$E := 29007547.53 \cdot psi$ $\quad F := 180000 \cdot lbf$

$L := 12 \cdot in$ $\quad b := 3 \cdot in$

$$m := \frac{3 \cdot in}{36 \cdot in} = 0.083$$

$$x := \begin{pmatrix} 4.5 \\ 13.5 \\ 22.5 \\ 31.5 \end{pmatrix} \cdot in$$ node x coordinates

$$h(x) := 2 \cdot (6 \cdot in - m \cdot x)$$

$$h(x) = \begin{pmatrix} 11.25 \\ 9.75 \\ 8.25 \\ 6.75 \end{pmatrix} in$$

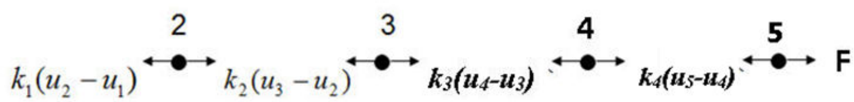
CALCULATIONS

Step 1. define axes and sign conventions. shown in figure above
Step 2. computer numerical values of k's:

$$A := b \cdot h(x) = \begin{pmatrix} 33.75 \\ 29.25 \\ 24.75 \\ 20.25 \end{pmatrix} in^2$$

$$k := \frac{E \cdot A}{L} = \begin{pmatrix} 8.158 \times 10^7 \\ 7.071 \times 10^7 \\ 5.983 \times 10^7 \\ 4.895 \times 10^7 \end{pmatrix} \cdot \frac{lbf}{in}$$

2.3. Draw free-body diagrams for the nodes where displacements are unknown for the tapered bar model described in Exercise 2.1.



2.4. Generate the nodal-force equilibrium equations at nodes where the displacements are unknown for the tapered-bar model described in Exercise 2.1.

$$-k_1\cdot(u_2 - u_1) + k_2\cdot(u_3 - u_2) = 0 \qquad 0 = (k_1 + k_2)\cdot u_2 - k_2\cdot u_3 \quad \text{where u1 = 0}$$

$$-k_2\cdot(u_3 - u_2) + k_3\cdot(u_4 - u_3) = 0 \qquad 0 = -k_2\cdot u_2 + (k_2 + k_3)\cdot u_3 - k_3\cdot u_4$$

$$-k_3\cdot(u_4 - u_3) + k_4\cdot(u_5 - u_4) = 0 \qquad 0 = -k_3\cdot u_3 + (k_3 + k_4)\cdot u_4 - k_4\cdot u_5$$

$$-k_4\cdot(u_5 - u_4) + F = 0 \qquad F = -k_4\cdot u_4 + k_4\cdot u_5$$

2.5. Develop the stiffness matrix, [K], and the force vector, {f}, for the tapered-bar model described in Exercise 2.1.

$$K := \begin{pmatrix} k_1 + k_2 & -k_2 & 0 & 0 \\ -k_2 & k_2 + k_3 & -k_3 & 0 \\ 0 & -k_3 & k_3 + k_4 & -k_4 \\ 0 & 0 & -k_4 & k_4 \end{pmatrix} = \begin{pmatrix} 2.031 \times 10^8 & -9.427 \times 10^7 & 0 & 0 \\ -9.427 \times 10^7 & 1.74 \times 10^8 & -7.977 \times 10^7 & 0 \\ 0 & -7.977 \times 10^7 & 1.45 \times 10^8 & -6.527 \times 10^7 \\ 0 & 0 & -6.527 \times 10^7 & 6.527 \times 10^7 \end{pmatrix} \frac{\text{lbf}}{\text{in}}$$

2.6. Solve the matrix equation for {u} for the tapered-bar model described in Exercise 2.1.

$$U := K^{-1} \cdot f \quad U = \begin{pmatrix} 1.655 \times 10^{-3} \\ 3.564 \times 10^{-3} \\ 5.821 \times 10^{-3} \\ 8.578 \times 10^{-3} \end{pmatrix} \text{in}$$

$$u := \begin{pmatrix} 0 \\ U_1 \\ U_2 \\ U_3 \\ U_4 \end{pmatrix} = \begin{pmatrix} 0 \\ 1.655 \times 10^{-3} \\ 3.564 \times 10^{-3} \\ 5.821 \times 10^{-3} \\ 8.578 \times 10^{-3} \end{pmatrix} \text{in}$$

2.7. Compute the element strains and stresses for the tapered-bar model described in Exercise 2.1.

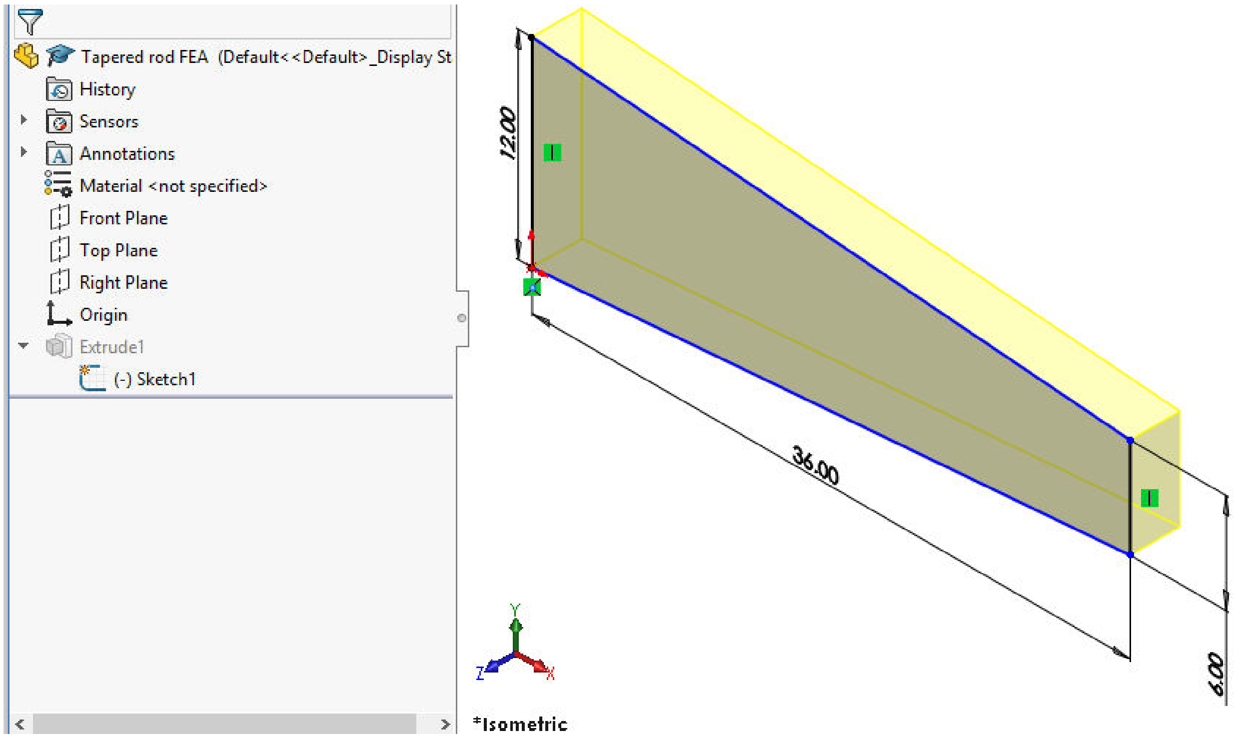
$$\delta := \begin{pmatrix} u_2 - u_1 \\ u_3 - u_2 \\ u_4 - u_3 \\ u_5 - u_4 \end{pmatrix} = \begin{pmatrix} 1.655 \times 10^{-3} \\ 1.909 \times 10^{-3} \\ 2.256 \times 10^{-3} \\ 2.758 \times 10^{-3} \end{pmatrix} \text{in}$$

$$\varepsilon := \frac{\delta}{L} = \begin{pmatrix} 1.839 \times 10^{-4} \\ 2.121 \times 10^{-4} \\ 2.507 \times 10^{-4} \\ 3.064 \times 10^{-4} \end{pmatrix}$$

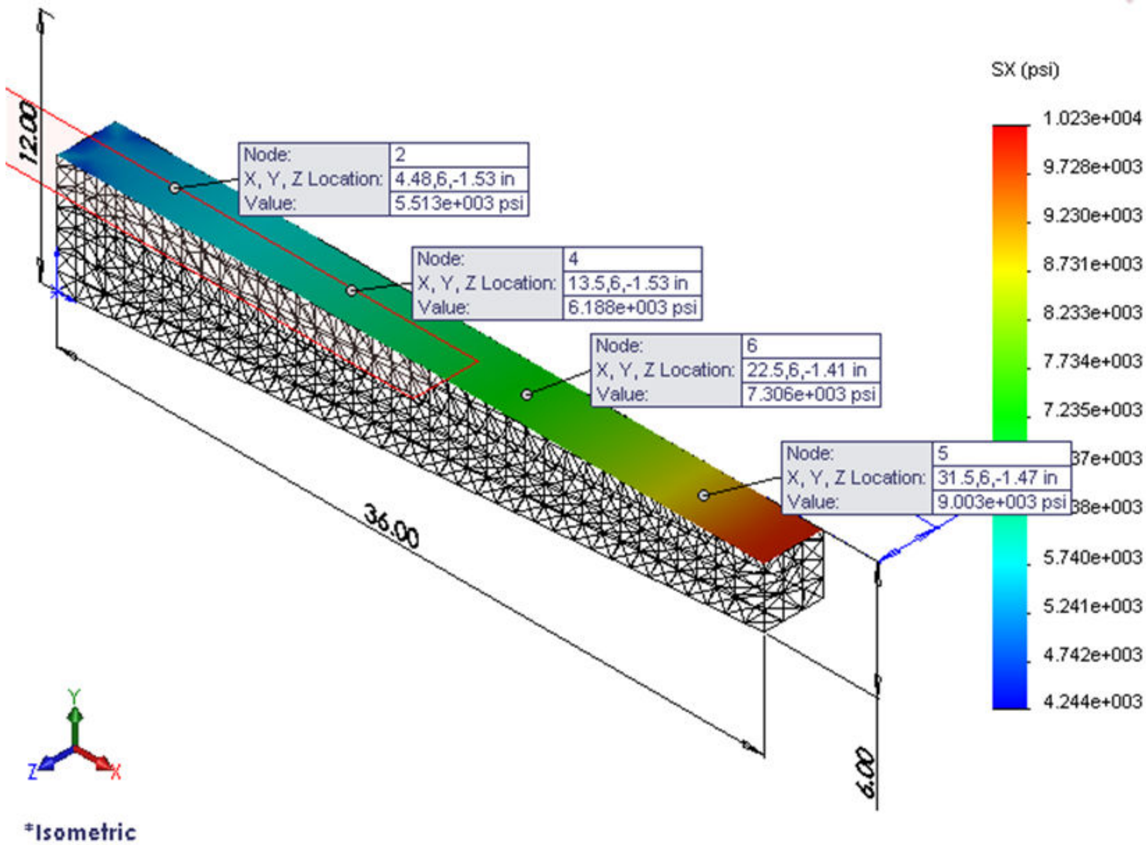
$$\sigma := \varepsilon \cdot E = \begin{pmatrix} 5.333 \times 10^3 \\ 6.154 \times 10^3 \\ 7.273 \times 10^3 \\ 8.889 \times 10^3 \end{pmatrix} psi$$

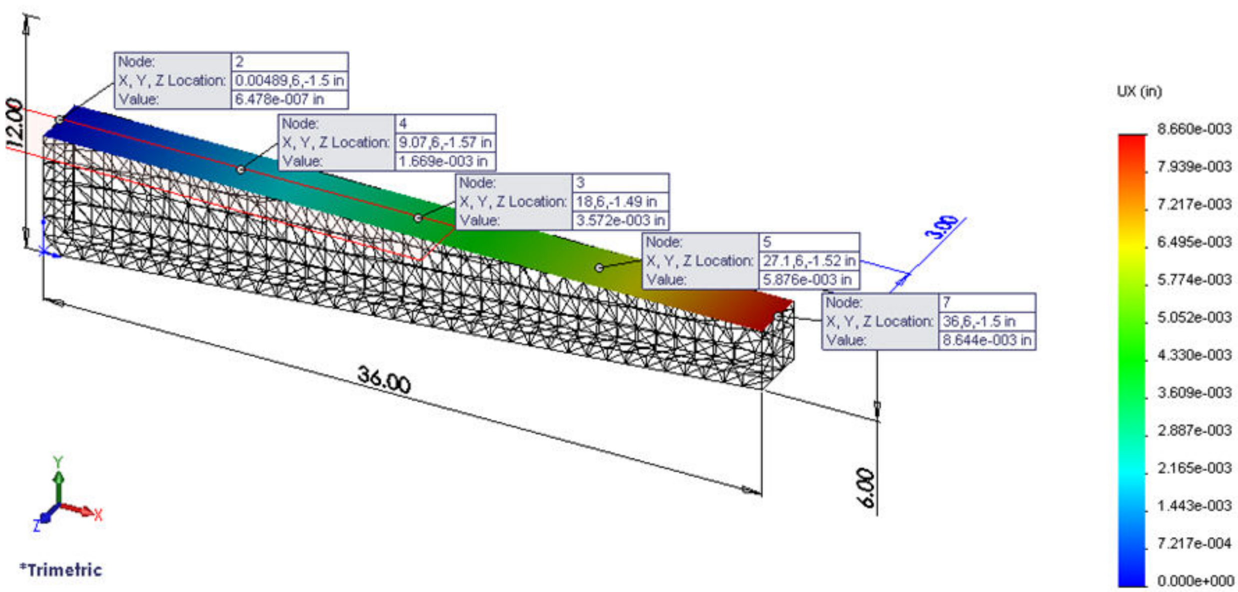
2.8. Create a 3D solid model of the tapered bar described in Exercise 2.1.

Sketch the Front Plane view and extrude the thickness.



2.9. Refer to Exercise 2.1. Conduct an FEA and create a contour plot of the x-normal stress and displacement. Create and probe a section clipping at the nodes. Compare the FEA results with the spring-element model results. Explain any differences.

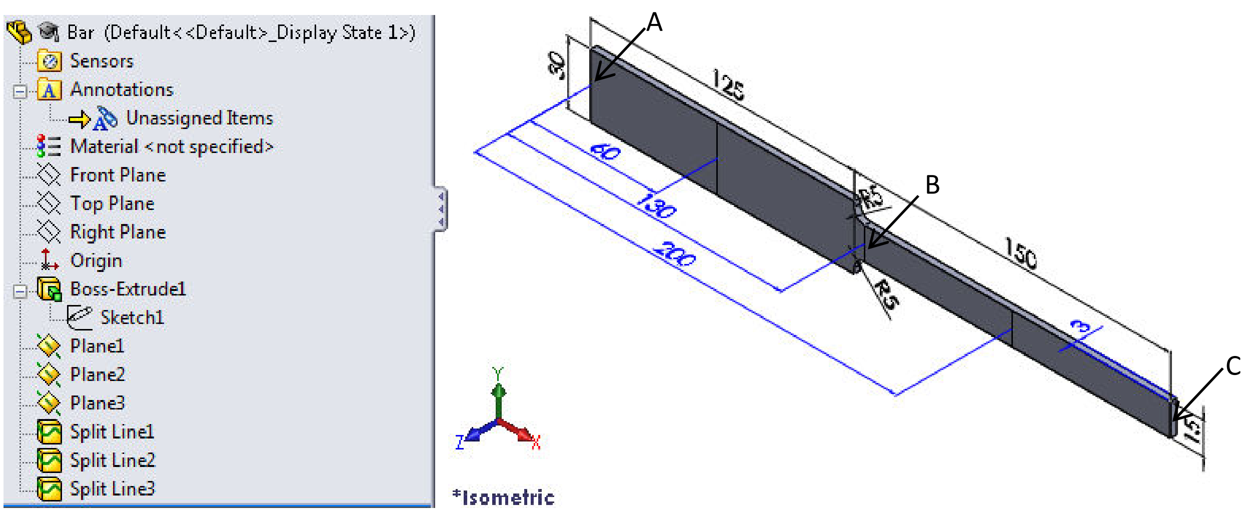

SX (psi)
1.023e+004
9.728e+003
9.230e+003
8.731e+003
8.233e+003
7.734e+003
7.235e+003
5.740e+003
5.241e+003
4.742e+003
4.244e+003
Node: 2
X, Y, Z Location: 4.48,6,-1.53 in
Value: 5.513e+003 psi
Node: 4
X, Y, Z Location: 13.5,6,-1.53 in
Value: 6.188e+003 psi
Node: 6
X, Y, Z Location: 22.5,6,-1.41 in
Value: 7.306e+003 psi
Node: 5
X, Y, Z Location: 31.5,6,-1.47 in
Value: 9.003e+003 psi
12.00
36.00
6.00
Y
Z
X
*Isometric


UX (in)
8.660e-003
7.939e-003
7.217e-003
6.495e-003
5.774e-003
5.052e-003
4.330e-003
3.609e-003
2.887e-003
2.165e-003
1.443e-003
7.217e-004
0.000e+000
Node: 2
X, Y, Z Location: 0.00489,6,-1.5 in
Value: 6.478e-007 in
Node: 4
X, Y, Z Location: 9.07,6,-1.57 in
Value: 1.669e-003 in
Node: 3
X, Y, Z Location: 18,6,-1.49 in
Value: 3.572e-003 in
Node: 5
X, Y, Z Location: 27.1,6,-1.52 in
Value: 5.876e-003 in
Node: 7
X, Y, Z Location: 36,6,-1.5 in
Value: 8.644e-003 in
12.00
3.00
36.00
6.00
Y
Z
X
*Trimetric

	Mcad	SWS
u_1	0	6.478e-7 in
u_2	1.655e-3 in	1.669e-3 in
u_3	3.564e-3 in	3.572e-3 in
u_4	5.821e-3 in	5.876e-3 in
u_5	8.578e-3 in	8.644e-3 in
σ_1	5333 psi	5513 psi
σ_2	6154 psi	6188 psi
σ_3	7273 psi	7306 psi
σ_4	8889 psi	9003 psi

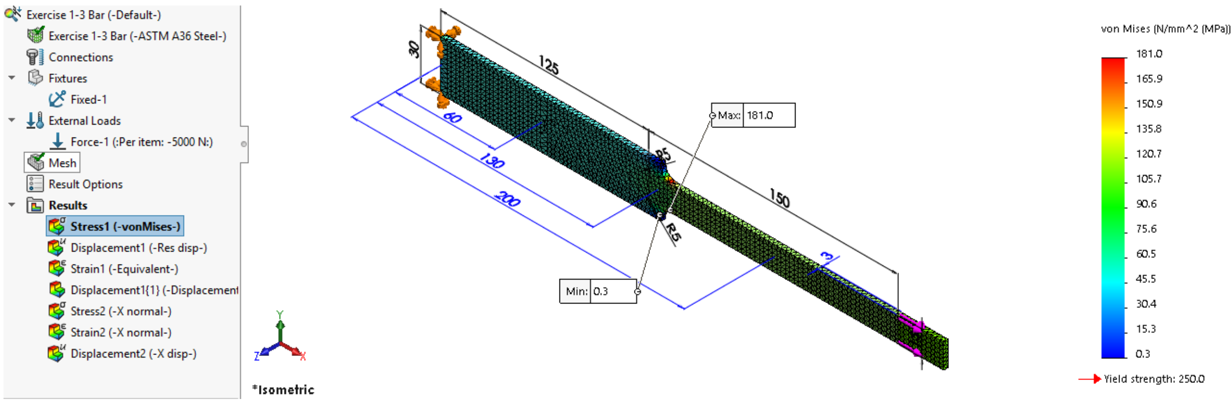
2.10. Create the 3D solid model shown. Use MMGS dimensions and SI units.

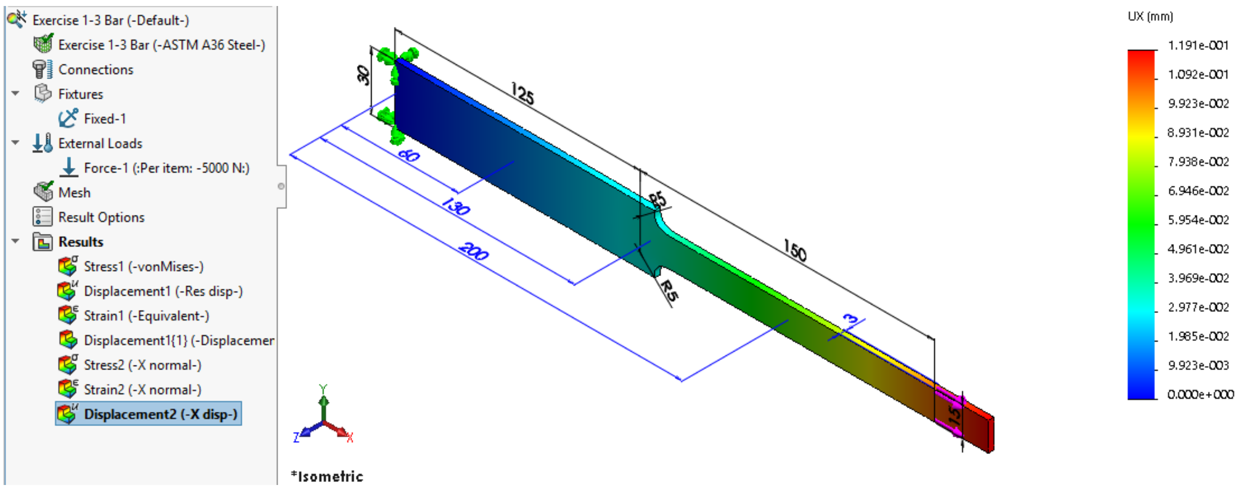
This is the same model as Exercise 1.3. All dimensions are in mm.



2.11. Create a 1D spring-element model for the 3D solid model from Exercise 2.10. Use ASTM A36 steel material, a fixed geometry restraint on the left-end face, a 5000 N load applied uniformly over the right-end face in the −y direction, and the default mesh. Create von Mises (vM) stress and x-displacement plots.

Create new, or use the vM-stress and x-displacement plots from Exercise 1.3.b:





The 1D spring model development is the same as in Exercises 2.1 – 2.7, but it is simpler because it only has two springs, one for the 30-mm height bar section and the other for the 15-mm height section. Just modify the drawings and the MCAD worksheet for a two spring model.

2.12. Refer to Exercise 2.10. Compare with FEA and *x*-normal stress calculations using mechanics of materials fundamentals in segment AB at *x* section 60 mm from the left end (split line 1). Identify all stresses as compressive (negative) or tensile (positive). Compute the percent difference with the FEA results.

Create new or use the MOM calculations from exercise 1.3.e:

$h_1 := 30\ mm \qquad h_2 := 15\ mm \qquad t := 3\ mm$

$A_1 := h_1 \cdot t = 90\ mm^2$

$A_2 := h_2 \cdot t = 45\ mm^2$

$F := 5000 \cdot N \qquad L := 1000 \cdot mm$

$\sigma_{x1} := \dfrac{F}{A_1} = 55.556\ MPa$

$\sigma_{x2} := \dfrac{F}{A_2} = 111.111\ MPa$

The average values of x-normal stress from the probe plots in Exercise 1.3.e were:

$\sigma_{x1SWS} := 58.585\ MPa \qquad \sigma_{x2SWS} := 111.1\ MPa$

These should compare favorably with the results from the spring element model.

2.13. Refer to Exercise 2.10. Use mechanics of materials fundamentals to calculate the deformation δ_{AC} in the x direction and compute the percent difference with the FEA results.

Calculate the displacement in the spring. Calculate or use the results from Exercise 1.3.f to get the MOM deformation:

$E := 200000 \cdot MPa \qquad L_1 := 125 \cdot mm \qquad L_2 := 150 \cdot mm$

$\varepsilon_{x1} := \dfrac{\sigma_{x1}}{E} = 2.778 \cdot 10^{-4} \qquad \delta_{x1} := \varepsilon_{x1} \cdot L_1 = 0.035\ mm$

$\varepsilon_{x2} := \dfrac{\sigma_{x2}}{E} = 5.556 \cdot 10^{-4} \qquad \delta_{x2} := \varepsilon_{x2} \cdot L_2 = 0.083\ mm$

$\delta_{x2} := \delta_{x1} + \delta_{x2} = 0.118\ mm$

Get Ux from Exercise 1.3.b or 2.11. They should compare favorably with the spring model.

2.14. Refer to Exercise 2.10. Calculate the x-normal stress using mechanics of materials (MOM) segment BC at the x section 200 mm from the left end (split line 3). Compute the percent difference with the FEA results.

The answer was given previously in Exercise 2.12.

2.15. Refer to Exercise 2.10. Use mechanics of materials fundamentals to calculate the *x* direction deformation in AC. Compute the percent difference with the FEA results.

The answer was given previously in Exercise 2.13. Add the displacements from the two springs to get the total deformation.

2.16. Develop a 1D spring-element model for an extruded aluminum strut from the item shown.

This is the same model as in Exercise 1.5. Students can reuse it if they built it previously.

Create the 3D solid model by downloading the cross-section profile shown and extruding it the distance shown. The origin is at the center of the cross-section sketch. Do not use a weldment profile as that automatically will create beam elements instead of 3D tetrahedral elements. Extrude the cross section so L = 1200 mm. Points A and B are at the left end, points A' and B' are at the right end. Points C, D, E, F, and G are at 120 mm from the left end.

Coordinates of points:
A = (0.0, 60.0, -8.0) mm,   A' = (1200.0, 60.0, -8.0) mm
B = (0.0, 38.0, 30.0) mm,   B' = (1200.0, 38.0, 30.0) mm
C = (120.0, 60.0, -8.0) mm,
D = (120.0, 30.0, -8.0) mm,
E = (120.0, 0.0, -26.5) mm,
F = (120.0, -30.0, -8.0) mm,
G = (120.0, -60.0, -8.0) mm.

Create the 1D spring element model with the same procedure as Exercises 2.1 – 2.7. This is a very simple spring model because the strut is uniform throughout its length and only one spring element is required.

2.17. Create a static study of the strut model created in Exercise 2.16. Use 6063-T6 aluminum alloy material. Apply a 1000 N external load in the negative y direction uniformly distributed on the right-end face. Apply a fixed geometry restraint to the left-end face (the face at the origin). Run the study with the default mesh configuration. Create a contour plot of *UY* in mm. Probe it along A–A′ on the upper-back corner of the strut.

See the answer for 1.5.b. This is the same analysis as Exercise 1.5.b. Students can reuse it if they completed it previously.

2.18. Refer to Exercise 2.16. To verify the FEA model configuration and to better understand the deformation probe plot shape and amplitude, use mechanics of materials fundamentals to create a graph of Uy(x) from A–A′.

See the answer for 1.5.c. This is the same graph as in Exercise 1.5.b. Students can reuse it if they completed it previously.

2.19. Refer to Exercise 2.16. Compare the FEA results for U_y @ x = 0 mm and U_y @ x = 1200 mm with the mechanics of materials fundamentals and the spring model results.
a. Explain any significant difference.
b. Is the probe plot of U_y linear? Explain.
c. Does the probe plot of U_y vary with x (distance from the left end)? Explain.

See the answer for 1.5.d. Since there is only one element in the spring model, it only determines U_y in two locations whereas the SWS and MCAD models show the displacement throughout the length of the strut. The SWS model uses many more elements, so even though it is a discrete result, it appears continuous. You can increase the resolution of the spring model by adding more elements (springs).

2.20. Refer to Exercise 2.16. Create a contour plot of the x-normal FEA stress results. Probe it along A–A′ on the upper-back corner of the strut. Notice St. Venant's effect on stress at the left end fixture. At what value of x does the St. Venant's effect first become negligible? Compare with the spring model results.

See the answer for 1.5.e. The simple single-element 1D spring model does not have the ability to determine St. Venant's effect.

2.21. Refer to Exercise 2.16. Using the x-normal stress results plot from FEA, create a section clipping plot 120 mm from the left end to avoid St. Venant's effect at the fixture. Probe at points C, D, E, F, and G. Coordinates of C = (120.0, 60.0, -8.0) mm, D = (120.0, 30.0, -8.0) mm, E = (120.0, 0.0, -26.5) mm, F = (120.0, -30.0, -8.0) mm, and G = (120.0, -60.0, -8.0) mm.

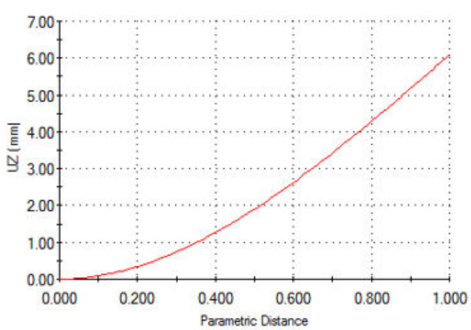
See the answer for 1.5.f. This is the same probe plot as Exercise 1.5.f. Students can reuse it if they completed it previously.

2.22. Refer to Exercise 2.16. To verify the FEA model configuration and to better understand the x-normal stress section clipping-plot probe values, use mechanics of materials fundamentals to calculate the normal stress values at points A', D, and F. Compare the values with the FEA results from Exercise 2.21 and the spring model results. Explain any significant differences.

See the answer for 1.5.g. This is the same MOM calculation as Exercise 1.5.g. Students can reuse it if they completed it previously. Since this spring model only has one dimension, it can't calculate stresses from a bending load. A 2D model will be introduced in Chapter 3.

2.23. Refer to Exercise 2.16. Change the applied load to 1000 N in the positive z direction uniformly distributed on the right-end face. Maintain the remaining configuration items, like the fixed geometry restraint on the left-end face and the default mesh. Create a contour plot of Uz(x) in mm. Probe it along B–B′ on the upper-back corner of the strut and create a probe plot of Uz. Coordinates are B = (0.0, 38.0, -30.0) mm and B′ = (1200.0, 38.0, 30.0) mm.

See the answer for 1.5.j. This is the same probe plot as Exercise 1.5.j. Students can reuse it if they completed it previously.



2.24. Refer to Exercises 2.16 and 2.23. To verify the FEA model configuration and to better understand the deformation probe plot shape and amplitude, use mechanics of materials fundamentals to create a graph of Uz(x) from B–B′. Compare with the FEA results probe plot and explain any differences. Compare the values of U_z @ x = 0 mm and U_z @ x = 1200 mm with the FEA results and the spring model and explain any differences.
a. Is the probe plot of U_z linear? Explain.

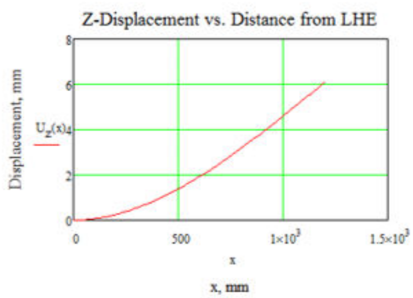
See the answer for 1.5.k. This is the same MOM plot as Exercise 1.5.g. Students can reuse it if they completed it previously.

No. The equation for δ is not linear.

$I_y := 1368425.30\text{mm}^4 \qquad F_z := 1000\text{N}$

$\delta_z(x) := -F_z \cdot x^2 \cdot \frac{(x - 3 \cdot L)}{6 \cdot E \cdot I_y}$

$\delta_z(1200\text{mm}) = 6.1\,\text{mm} \qquad U_z(x) := 0\text{m} + \delta_z(x) \qquad U_z(1200\text{mm}) = 6.1\,\text{mm}$



b. Is the deformation in the z direction of the Z-load model significantly different from the deformation in y direction of the Y-load model? Explain.

Yes. Deflection is inversely proportional to the area moment of inertia, which is different with respect to the y- and z-axes.

2.25. Is the FEA process described in this chapter valid outside of the linear stress–strain relationship of a material? For example, is it valid if the stress exceeds the yield point? Explain.

No. The spring-element model only simulates linear relationships.