Chapter 2

Bars and Trusses

Introduction

- Trusses are commonly used in the design of buildings, bridges and towers.
- This chapter introduces you to the simplest 1-D structural element, the bar element, and the finite element analysis of truss structures using such element.

(a)

(b)

Figure 2.1. Truss examples: (a) Montreal Biosphere Museum (b) Betsy Ross Bridge.

Introduction

- Most structural analysis problems can be treated as linear static problem, based on the following assumptions

  - *Small deformations* (loading pattern is not changed due to the deformed shape)
  - *Elastic materials* (no plasticity or failures)
  - *Static loads* (the load is applied to the structure in a slow or steady fashion)

- Linear analysis can provide most of the information about the behavior of a structure, and can be a good approximation for many analyses.

Review of the 1-D Elasticity Theory

- Consider a uniform prismatic bar shown below

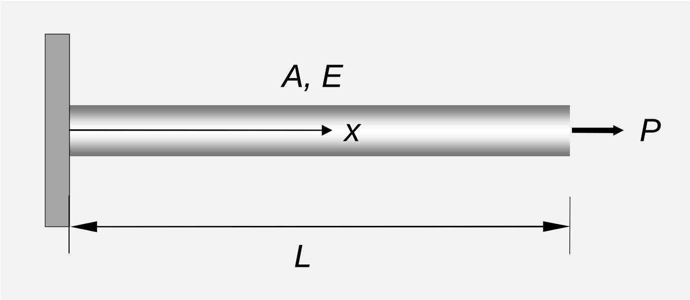


Figure 2.2. An axially loaded elastic bar.

Strain-displacement relation

$$\varepsilon(x) = \frac{du(x)}{dx}$$

Strain-displacement relation

$$\sigma(x) = E\,\varepsilon(x)$$

Equilibrium equation

$$\frac{d\sigma(x)}{dx} + f(x) = 0$$

The displacement, strain and stress field in a bar needs to be solved under given boundary conditions, which can be done readily for a single bar, but can be tedious for a network of bars or a truss structure made of many bars.

A truss is an assembly of axial bars ...

Truss Modeling & Bar Element Formulation

Modeling of Trusses

- For the truss analysis, it is often assumed that
  - The bars are of uniform cross sections and joined by frictionless pins.
  - Loads are applied to joints only.
- Based on the assumptions, truss members are considered to carry only axial loads and have negligible bending resistance.

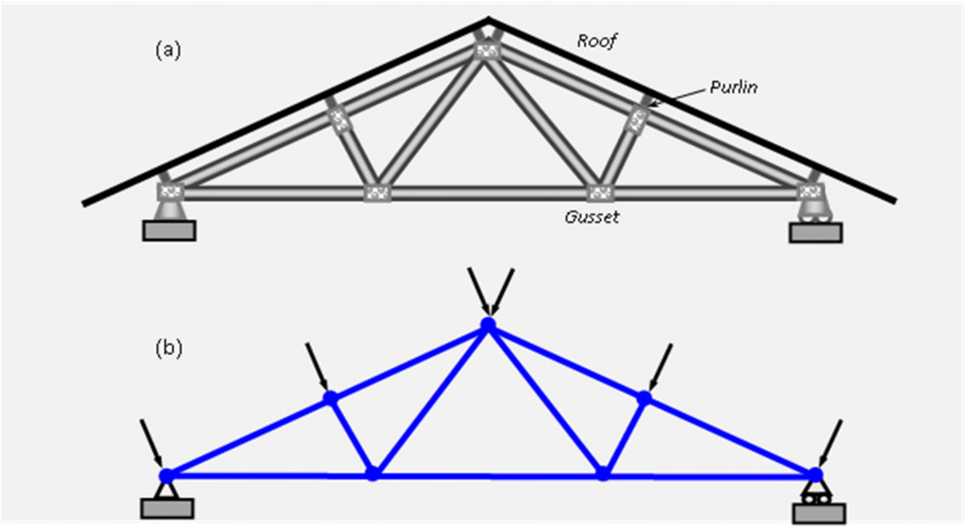


Figure 2.3. Modeling of a planar roof truss: (a) Physical structure (b) Discrete model.

Formulation of the Bar Element

❑ Stiffness Matrix – Direct Method

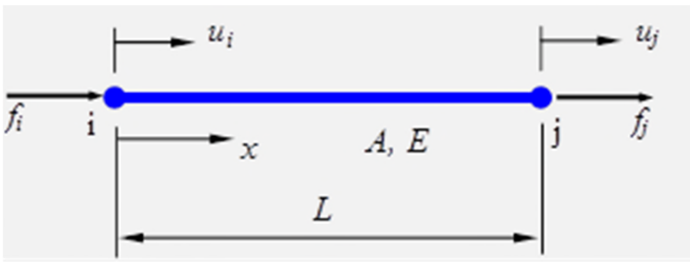


Figure 2.4. Notation for a bar element.

We have $u(x)=\left(1-\frac{x}{L}\right)u_i+\frac{x}{L}u_j$

$$\varepsilon=\frac{u_j-u_i}{L}=\frac{\Delta}{L}$$

$$\sigma=E\varepsilon=\frac{E\Delta}{L} \text{ and } \sigma=\frac{F}{A}$$

Therefore $F=\frac{EA}{L}\Delta=k\Delta$

We conclude that the bar behaves like a spring. The element stiffness matrix is:

$$\mathbf{k}=\begin{bmatrix} k & -k \\ -k & k \end{bmatrix}=\begin{bmatrix} \frac{EA}{L} & -\frac{EA}{L} \\ -\frac{EA}{L} & \frac{EA}{L} \end{bmatrix}$$

Element equilibrium equation is:

$$\frac{EA}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} u_i \\ u_j \end{Bmatrix}=\begin{Bmatrix} f_i \\ f_j \end{Bmatrix}$$

Degree of Freedom (DOF): Number of components of the displacement vector at a node. For 1-D bar element along the *x*-axis, we have one DOF at each node.

Formulation of the Bar Element

❑ Stiffness Matrix – Energy Approach

Define two *linear shape functions* as follows

$$N_i(\xi) = 1 - \xi, \qquad N_j(\xi) = \xi \quad \text{where} \quad \xi = \frac{x}{L}, \qquad 0 \le \xi \le 1$$

We can write

$$u = \begin{bmatrix} N_i & N_j \end{bmatrix} \begin{Bmatrix} u_i \\ u_j \end{Bmatrix} = \mathbf{Nu} \qquad \text{and} \qquad \varepsilon = \frac{du}{dx} = \left[\frac{d}{dx} \mathbf{N} \right] \mathbf{u} = \mathbf{Bu}$$

where **B** is the element *strain-displacement matrix* $\mathbf{B} = \begin{bmatrix} -1/L & 1/L \end{bmatrix}$

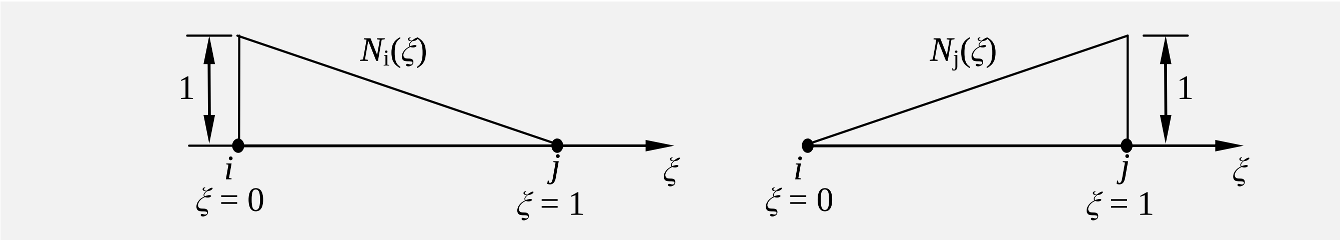


Figure 2.5. The shape functions for a bar element.

Formulation of the Bar Element

Stress can be written as

$$\sigma = E\varepsilon = E\mathbf{B}\mathbf{u}$$

Consider the stored *strain energy*

$$U = \frac{1}{2}\int_V \sigma^{\mathrm{T}}\varepsilon dV = \frac{1}{2}\int_V \left(\mathbf{u}^{\mathrm{T}}\mathbf{B}^{\mathrm{T}}E\mathbf{B}\mathbf{u}\right)dV$$

$$= \frac{1}{2}\mathbf{u}^{\mathrm{T}}\left[\int_V \left(\mathbf{B}^{\mathrm{T}}E\mathbf{B}\right)dV\right]\mathbf{u}$$

The *potential* of the external forces is

$$\Omega = -f_i u_i - f_j u_j = -\mathbf{u}^{\mathrm{T}}\mathbf{f}$$

The total potential of the system is

$$\Pi = U + \Omega$$

$$\Pi = \frac{1}{2}\mathbf{u}^{\mathrm{T}}\left[\int_V \left(\mathbf{B}^{\mathrm{T}}E\mathbf{B}\right)dV\right]\mathbf{u} - \mathbf{u}^{\mathrm{T}}\mathbf{f}$$

Setting $d\Pi = 0$ by the principle of minimum potential energy, we obtain

$$\mathbf{k}\mathbf{u} = \mathbf{f}$$

where $\mathbf{k} = \int_V \left(\mathbf{B}^{\mathrm{T}}E\mathbf{B}\right)dV$

For the bar element

$$\mathbf{k} = \int_0^L \begin{Bmatrix} -1/L \\ 1/L \end{Bmatrix} E\begin{bmatrix} -1/L & 1/L \end{bmatrix} A dx = \frac{EA}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$

Formulation of the Bar Element

❑ Treatment of Distributed Load

Distributed axial load *q* can be converted to two equivalent nodal forces using the shape functions. Consider the work done by the distributed load *q.*

$$W_q = \frac{1}{2}\int_0^L u(x)q(x)dx = \frac{1}{2}\int_0^L (\mathbf{N}\mathbf{u})^T q(x)dx = \frac{1}{2}\begin{bmatrix} u_i & u_j \end{bmatrix}\int_0^L \begin{bmatrix} N_i(x) \\ N_j(x) \end{bmatrix} q(x)dx$$

$$= \frac{1}{2}\mathbf{u}^T \int_0^L \mathbf{N}^T q(x)dx$$

The work done by the equivalent nodal forces are

$$W_{f_q} = \frac{1}{2} f_i^q u_i + \frac{1}{2} f_j^q u_j = \frac{1}{2}\mathbf{u}^T \mathbf{f}_q$$

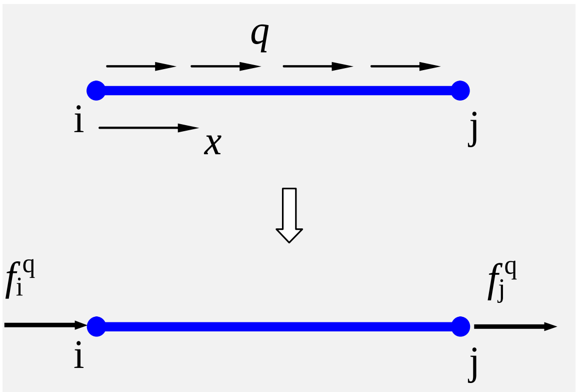


Figure 2.6. Conversion of a distributed load on one element.

Formulation of the Bar Element

Setting $W_q = W_{f_q}$, we obtain the equivalent nodal force vector

$$\mathbf{f}_q = \begin{Bmatrix} f_i^{\,q} \\ f_j^{\,q} \end{Bmatrix} = \int_0^L \mathbf{N}^T q(x)dx = \int_0^L \begin{bmatrix} N_i(x) \\ N_j(x) \end{bmatrix} q(x)dx$$

If *q* is a *constant*, we have

$$\mathbf{f}_q = q\int_0^L \begin{bmatrix} 1 - x/L \\ x/L \end{bmatrix} dx = \begin{Bmatrix} qL/2 \\ qL/2 \end{Bmatrix}$$

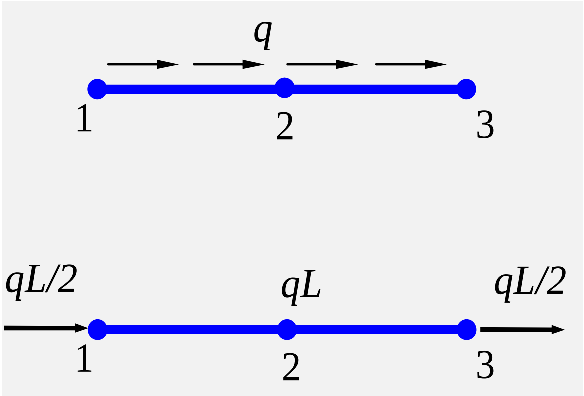


Figure 2.7. Conversion of a distributed load with constant intensity *q* on two elements.

Formulation of the Bar Element

- Bar Element In 2-D and 3-D

2-D Case

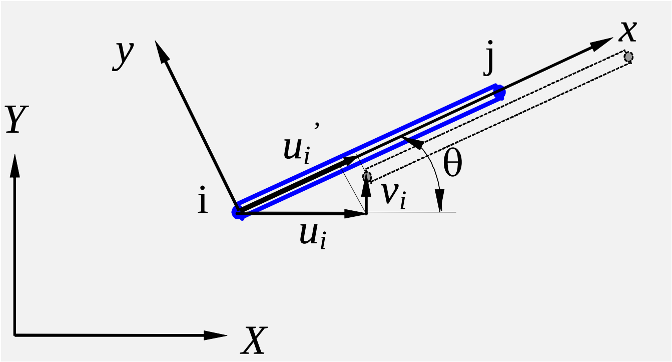


Figure 2.8. Local and global coordinates for a bar in 2-D space.

Local	*Global*
x, y u_i', v_i' 1 DOF at each node	X, Y u_i, v_i 2 DOFs at each node

Formulation of the Bar Element

Displacement vectors in the local and global coordinates are related as follows

$$u_i^{'} = u_i \cos\theta + v_i \sin\theta = \begin{bmatrix} l & m \end{bmatrix} \begin{Bmatrix} u_i \\ v_i \end{Bmatrix}$$

$$v_i^{'} = -u_i \sin\theta + v_i \cos\theta = \begin{bmatrix} -m & l \end{bmatrix} \begin{Bmatrix} u_i \\ v_i \end{Bmatrix}$$

where $l = \cos\theta, m = \sin\theta$

In matrix form

$$\begin{Bmatrix} u_i^{'} \\ v_i^{'} \end{Bmatrix} = \begin{bmatrix} l & m \\ -m & l \end{bmatrix} \begin{Bmatrix} u_i \\ v_i \end{Bmatrix}$$ or $$\mathbf{u}_i^{'} = \tilde{\mathbf{T}}\mathbf{u}_i$$

For the two nodes of the bar element, we have

$$\begin{Bmatrix} u_i^{'} \\ v_i^{'} \\ u_j^{'} \\ v_j^{'} \end{Bmatrix} = \begin{bmatrix} l & m & 0 & 0 \\ -m & l & 0 & 0 \\ 0 & 0 & l & m \\ 0 & 0 & -m & l \end{bmatrix} \begin{Bmatrix} u_i \\ v_i \\ u_j \\ v_j \end{Bmatrix}$$

In matrix form

$$\mathbf{u}^{'} = \mathbf{T}\mathbf{u}$$ with $$\mathbf{T} = \begin{bmatrix} \tilde{\mathbf{T}} & \mathbf{0} \\ \mathbf{0} & \tilde{\mathbf{T}} \end{bmatrix}$$

Formulation of the Bar Element

In the local coordinate system, we have

$$\frac{EA}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} u_i^{'} \\ u_j^{'} \end{Bmatrix} = \begin{Bmatrix} f_i^{'} \\ f_j^{'} \end{Bmatrix}$$

Augmenting this equation, we write

$$\frac{EA}{L}\begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix}\begin{Bmatrix} u_i^{'} \\ v_i^{'} \\ u_j^{'} \\ v_j^{'} \end{Bmatrix} = \begin{Bmatrix} f_i^{'} \\ 0 \\ f_j^{'} \\ 0 \end{Bmatrix}$$

or $\mathbf{k}^{'}\mathbf{u}^{'} = \mathbf{f}^{'}$

Formulation of the Bar Element

Using the transformations, we obtain

$$\mathbf{k}'\mathbf{T}\mathbf{u} = \mathbf{T}\mathbf{f}$$

$$\mathbf{T}^T\mathbf{k}'\mathbf{T}\mathbf{u} = \mathbf{f}$$

Thus, the element stiffness matrix **k** in the global coordinate system is

$$\mathbf{k} = \mathbf{T}^T\mathbf{k}'\mathbf{T}$$

The explicit form

$$\mathbf{k} = \frac{EA}{L}\begin{array}{c} \begin{array}{cccc} u_i & v_i & u_j & v_j \end{array} \\ \begin{bmatrix} l^2 & lm & -l^2 & -lm \\ lm & m^2 & -lm & -m^2 \\ -l^2 & -lm & l^2 & lm \\ -lm & -m^2 & lm & m^2 \end{bmatrix} \end{array}$$

Formulation of the Bar Element

3-D Case

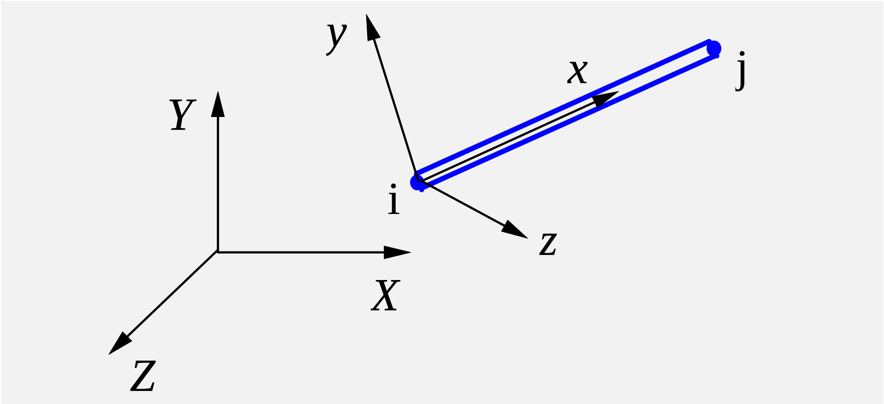


Figure 2.9. Local and global coordinates for a bar in 3-D space.

Local	*Global*
x, y, z	X, Y, Z
u_i', v_i', w_i'	u_i, v_i, w_i
1 DOF at each node	3 DOFs at each node

Formulation of the Bar Element

The transformation relation is

$$\begin{Bmatrix} u_i' \\ v_i' \\ w_i' \end{Bmatrix} = \begin{bmatrix} l_X & l_Y & l_Z \\ m_X & m_Y & m_Z \\ n_X & n_Y & n_Z \end{bmatrix} \begin{Bmatrix} u_i \\ v_i \\ w_i \end{Bmatrix}$$

where (l_X, l_Y, l_Z), (m_X, m_Y, m_Z) and (n_X, n_Y, n_Z) are the direction cosines of the local *x*, *y* and *z* coordinate axis in the global coordinate system, respectively.

FEM software packages will do this transformation automatically.

The input data for bar elements are simply the coordinates (*X, Y, Z*) for each node, *E* and *A* for each element (Length *L* can be computed from the coordinates of the two nodes).

Formulation of the Bar Element

❑ Element Stress

Once the nodal displacement is obtained for an element, the stress within the element can be calculated using the basic relations.

For 2-D cases, we proceed as follows

$$\sigma = E\varepsilon = E\mathbf{B}\begin{Bmatrix} u_i' \\ u_j' \end{Bmatrix} = E\begin{bmatrix} -\frac{1}{L} & \frac{1}{L} \end{bmatrix}\begin{bmatrix} l & m & 0 & 0 \\ 0 & 0 & l & m \end{bmatrix}\begin{Bmatrix} u_i \\ v_i \\ u_j \\ v_j \end{Bmatrix}$$

That is,

$$\sigma = \frac{E}{L}\begin{bmatrix} -l & -m & l & m \end{bmatrix}\begin{Bmatrix} u_i \\ v_i \\ u_j \\ v_j \end{Bmatrix}$$

which is a general formula for 2-D bar elements.

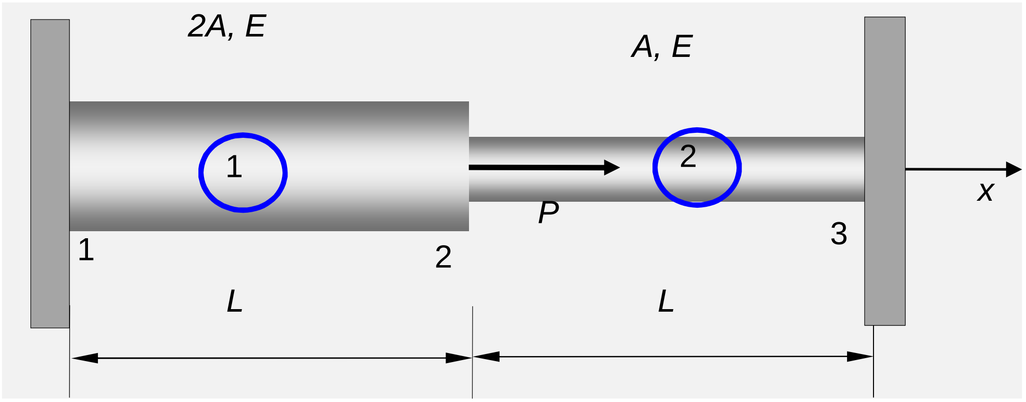
We now look at examples of 1-D stress and plane truss problems …

Examples with Bar Elements

Examples with Bar Elements

- Example Problems

Example 2.1



Problem: Find the stresses in the two-bar assembly which is loaded with force *P*, and constrained at the two ends.

Solution: Use two 1-D bar elements.

Examples with Bar Elements

For element 1,

$$\mathbf{k}_1 = \frac{2EA}{L}\begin{array}{cc} u_1 & u_2 \\ \end{array}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$

For element 2,

$$\mathbf{k}_2 = \frac{EA}{L}\begin{array}{cc} u_2 & u_3 \\ \end{array}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$

Assemble the global FE equation as follows

$$\frac{EA}{L}\begin{bmatrix} 2 & -2 & 0 \\ -2 & 3 & -1 \\ 0 & -1 & 1 \end{bmatrix}\begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \end{Bmatrix}$$

Examples with Bar Elements

Load and boundary conditions (BCs) are

$$u_1 = u_3 = 0, \qquad F_2 = P$$

FE equation becomes

$$\frac{EA}{L}\begin{bmatrix} 2 & -2 & 0 \\ -2 & 3 & -1 \\ 0 & -1 & 1 \end{bmatrix}\begin{Bmatrix} 0 \\ u_2 \\ 0 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ P \\ F_3 \end{Bmatrix}$$

Thus,

$$\begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \frac{PL}{3EA}\begin{Bmatrix} 0 \\ 1 \\ 0 \end{Bmatrix}$$

Examples with Bar Elements

Stress in element 1 is

$$\sigma_1 = E\varepsilon_1 = E\mathbf{B}_1\mathbf{u}_1 = E\begin{bmatrix}-1/L & 1/L\end{bmatrix}\begin{Bmatrix}u_1 \\ u_2\end{Bmatrix}$$

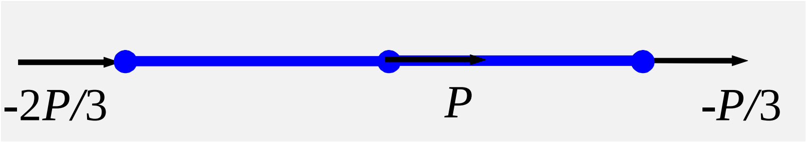
$$= E\frac{u_2 - u_1}{L} = \frac{E}{L}\left(\frac{PL}{3EA} - 0\right) = \frac{P}{3A}$$

Similarly, stress in element 2 is

$$\sigma_2 = E\varepsilon_2 = E\mathbf{B}_2\mathbf{u}_2 = E\begin{bmatrix}-1/L & 1/L\end{bmatrix}\begin{Bmatrix}u_2 \\ u_3\end{Bmatrix}$$

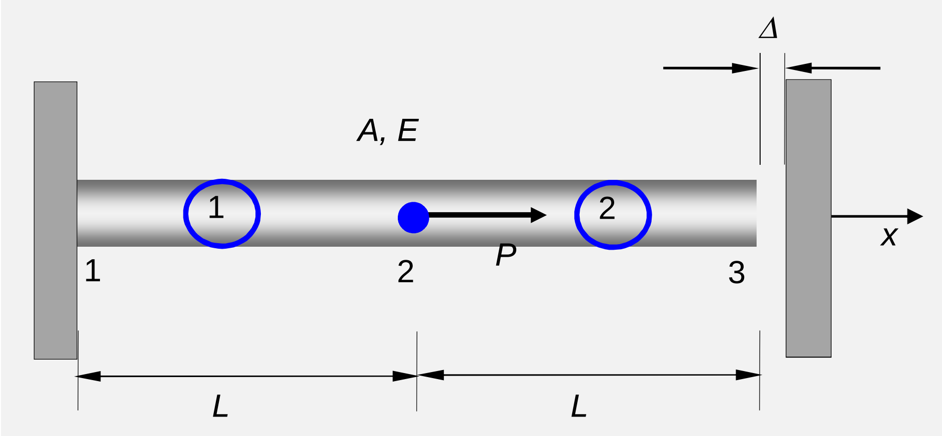
$$= E\frac{u_3 - u_2}{L} = \frac{E}{L}\left(0 - \frac{PL}{3EA}\right) = -\frac{P}{3A}$$

***Check the results*:** Draw the FBD and check the equilibrium of the structures.



Examples with Bar Elements

Example 2.2



Problem: Determine the support reaction forces at the two ends of the bar shown above, given the following

$$P = 6.0\times 10^4 \text{ N},\quad E = 2.0\times 10^4 \text{ N/mm}^2,$$
$$A = 250 \text{ mm}^2,\quad L = 150 \text{ mm},\quad \Delta = 1.2 \text{ mm}$$

Examples with Bar Elements

We first check to see if contact of the bar with the wall on the right will occur or not. To do this, we imagine the wall on the right is removed and calculate the displacement at the right end.

$$\Delta_0 = \frac{PL}{EA} = \frac{(6.0\times10^4)(150)}{(2.0\times10^4)(250)} = 1.8\text{mm} > \Delta = 1.2\text{mm}$$

Thus, contact occurs and the wall on the right should be accounted for in the analysis.

The global FE equation is found to be

$$\frac{EA}{L}\begin{bmatrix} 1 & -1 & 0 \\ -1 & 2 & -1 \\ 0 & -1 & 1 \end{bmatrix}\begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \end{Bmatrix}$$

Examples with Bar Elements

The load and boundary conditions are

$$F_2 = P = 6.0 \times 10^4 \,\mathrm{N}$$

$$u_1 = 0, \qquad\qquad u_3 = \Delta = 1.2\mathrm{mm}$$

FE equation becomes

$$\frac{EA}{L}\begin{bmatrix} 1 & -1 & 0 \\ -1 & 2 & -1 \\ 0 & -1 & 1 \end{bmatrix}\begin{Bmatrix} 0 \\ u_2 \\ \Delta \end{Bmatrix} = \begin{Bmatrix} F_1 \\ P \\ F_3 \end{Bmatrix}$$

Solving this, we obtain

$$\begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 1.5 \\ 1.2 \end{Bmatrix} (\mathrm{mm})$$

Examples with Bar Elements

To calculate the support reaction forces, we apply the 1st and 3rd equations in the global FE equation.

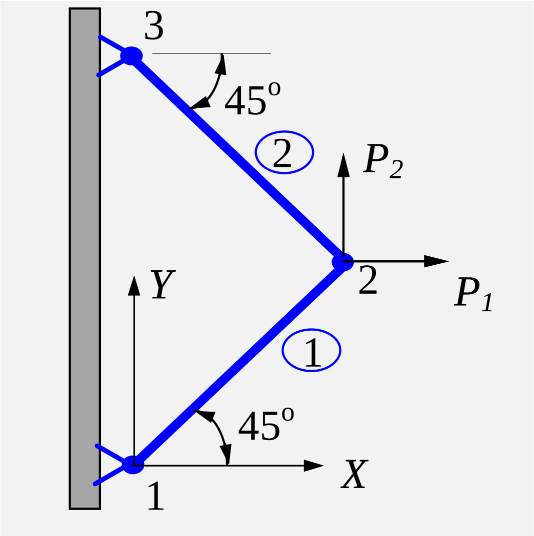
$$F_1 = \frac{EA}{L}\begin{bmatrix}1 & -1 & 0\end{bmatrix}\begin{Bmatrix}u_1 \\ u_2 \\ u_3\end{Bmatrix} = \frac{EA}{L}\left(-u_2\right) = -5.0\times 10^4\,\mathrm{N}$$

$$F_3 = \frac{EA}{L}\begin{bmatrix}0 & -1 & 1\end{bmatrix}\begin{Bmatrix}u_1 \\ u_2 \\ u_3\end{Bmatrix} = \frac{EA}{L}\left(-u_2 + u_3\right) = -1.0\times 10^4\,\mathrm{N}$$

Check the results!

Examples with Bar Elements

Example 2.3



A simple plane truss is made of two identical bars (with *E, A,* and *L*), and loaded as shown in the above figure.

Find:

(a) displacement of node 2;

(b) stress in each bar.

Examples with Bar Elements

In local coordinate systems, we have

$$\mathbf{k}_1' = \frac{EA}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = \mathbf{k}_2'$$

Element 1:

$$\theta = 45^o, \qquad l = m = \frac{\sqrt{2}}{2}$$

$$\mathbf{k}_1 = \mathbf{T}_1^T \mathbf{k}_1' \mathbf{T}_1 = \frac{EA}{2L}\begin{array}{c} \begin{array}{cccc} u_1 & v_1 & u_2 & v_2 \end{array} \\ \begin{bmatrix} 1 & 1 & -1 & -1 \\ 1 & 1 & -1 & -1 \\ -1 & -1 & 1 & 1 \\ -1 & -1 & 1 & 1 \end{bmatrix} \end{array}$$

Element 2:

$$\theta = 135^o, \qquad l = -\frac{\sqrt{2}}{2}, m = \frac{\sqrt{2}}{2}$$

$$\mathbf{k}_2 = \mathbf{T}_2^T \mathbf{k}_2' \mathbf{T}_2 = \frac{EA}{2L}\begin{array}{c} \begin{array}{cccc} u_2 & v_2 & u_3 & v_3 \end{array} \\ \begin{bmatrix} 1 & -1 & -1 & 1 \\ -1 & 1 & 1 & -1 \\ -1 & 1 & 1 & -1 \\ 1 & -1 & -1 & 1 \end{bmatrix} \end{array}$$

Examples with Bar Elements

Assemble the structure FE equation

$$\frac{EA}{2L}\begin{array}{c} \begin{matrix} u_1 & v_1 & u_2 & v_2 & u_3 & v_3 \end{matrix} \\ \begin{bmatrix} 1 & 1 & -1 & -1 & 0 & 0 \\ 1 & 1 & -1 & -1 & 0 & 0 \\ -1 & -1 & 2 & 0 & -1 & 1 \\ -1 & -1 & 0 & 2 & 1 & -1 \\ 0 & 0 & -1 & 1 & 1 & -1 \\ 0 & 0 & 1 & -1 & -1 & 1 \end{bmatrix} \end{array} \begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \end{Bmatrix} = \begin{Bmatrix} F_{1X} \\ F_{1Y} \\ F_{2X} \\ F_{2Y} \\ F_{3X} \\ F_{3Y} \end{Bmatrix}$$

Load and boundary conditions (BC)

$$u_1 = v_1 = u_3 = v_3 = 0, \quad F_{2X} = P_1, F_{2Y} = P_2$$

Condensed FE equation

$$\frac{EA}{2L}\begin{bmatrix} 2 & 0 \\ 0 & 2 \end{bmatrix}\begin{Bmatrix} u_2 \\ v_2 \end{Bmatrix} = \begin{Bmatrix} P_1 \\ P_2 \end{Bmatrix}$$

Examples with Bar Elements

Solving this, we obtain

$$\begin{Bmatrix} u_2 \\ v_2 \end{Bmatrix} = \frac{L}{EA} \begin{Bmatrix} P_1 \\ P_2 \end{Bmatrix}$$

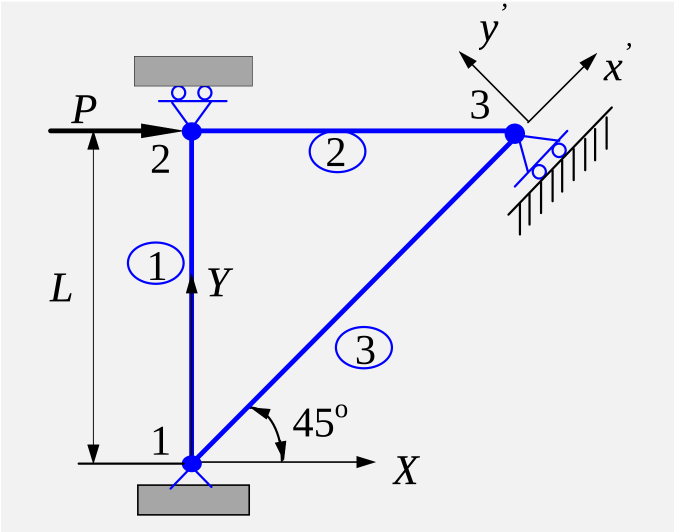
Stresses in the two bars

$$\sigma_1 = \frac{E}{L}\frac{\sqrt{2}}{2}\begin{bmatrix} -1 & -1 & 1 & 1 \end{bmatrix} \frac{L}{EA} \begin{Bmatrix} 0 \\ 0 \\ P_1 \\ P_2 \end{Bmatrix} = \frac{\sqrt{2}}{2A}\left(P_1 + P_2\right)$$

$$\sigma_2 = \frac{E}{L}\frac{\sqrt{2}}{2}\begin{bmatrix} 1 & -1 & -1 & 1 \end{bmatrix} \frac{L}{EA} \begin{Bmatrix} P_1 \\ P_2 \\ 0 \\ 0 \end{Bmatrix} = \frac{\sqrt{2}}{2A}\left(P_1 - P_2\right)$$

Examples with Bar Elements

Example 2.4 (Multipoint Constraint)



For the plane truss shown above,

$$P = 1000\ \text{kN}, \quad L = 1m,\ E = 210\,GPa,$$

$$A = 6.0 \times 10^{-4}\ m^2 \qquad \text{for elements } 1 \text{ and } 2,$$

$$A = 6\sqrt{2} \times 10^{-4}\ m^2 \qquad \text{for element } 3.$$

Determine the displacements and reaction forces.

Examples with Bar Elements

Element 1: $\theta = 90^o$, $l = 0, m = 1$

$$\mathbf{k}_1 = \frac{(210\times 10^9)(6.0\times 10^{-4})}{1} \begin{array}{c} \begin{array}{cccc} u_1 & v_1 & u_2 & v_2 \end{array} \\ \begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & -1 \\ 0 & 0 & 0 & 0 \\ 0 & -1 & 0 & 1 \end{bmatrix} \end{array} \text{(N/m)}$$

Element 2: $\theta = 0^o$, $l = 1, m = 0$

$$\mathbf{k}_2 = \frac{(210\times 10^9)(6.0\times 10^{-4})}{1} \begin{array}{c} \begin{array}{cccc} u_2 & v_2 & u_3 & v_3 \end{array} \\ \begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \end{array} \text{(N/m)}$$

Element 3: $\theta = 45^o$, $l = \frac{1}{\sqrt{2}}, m = \frac{1}{\sqrt{2}}$

$$\mathbf{k}_3 = \frac{(210\times 10^9)(6\sqrt{2}\times 10^{-4})}{\sqrt{2}} \begin{array}{c} \begin{array}{cccc} u_1 & v_1 & u_3 & v_3 \end{array} \\ \begin{bmatrix} 0.5 & 0.5 & -0.5 & -0.5 \\ 0.5 & 0.5 & -0.5 & -0.5 \\ -0.5 & -0.5 & 0.5 & 0.5 \\ -0.5 & -0.5 & 0.5 & 0.5 \end{bmatrix} \end{array} \text{(N/m)}$$

Examples with Bar Elements

The global FE equation is

$$1260\times10^{5}\begin{bmatrix} 0.5 & 0.5 & 0 & 0 & -0.5 & -0.5 \\ & 1.5 & 0 & -1 & -0.5 & -0.5 \\ & & 1 & 0 & -1 & 0 \\ & & & 1 & 0 & 0 \\ & & & & 1.5 & 0.5 \\ Sym. & & & & & 0.5 \end{bmatrix}\begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \end{Bmatrix}=\begin{Bmatrix} F_{1X} \\ F_{1Y} \\ F_{2X} \\ F_{2Y} \\ F_{3X} \\ F_{3Y} \end{Bmatrix}$$

Load and boundary conditions (BCs)

$$u_1 = v_1 = v_2 = 0, \text{and } v_3^{'} = 0,$$

$$F_{2X} = P, \qquad F_{3x'} = 0.$$

Examples with Bar Elements

From the transformation relation and the BC, we have

$$v_3^{'} = \begin{bmatrix} -\frac{\sqrt{2}}{2} & \frac{\sqrt{2}}{2} \end{bmatrix} \begin{Bmatrix} u_3 \\ v_3 \end{Bmatrix} = \frac{\sqrt{2}}{2}(-u_3 + v_3) = 0,$$

that is

$$u_3 - v_3 = 0$$

This is a *multipoint constraint* (MPC).

Similarly, we have a relation for the force at node 3

$$F_{3x'} = \begin{bmatrix} \frac{\sqrt{2}}{2} & \frac{\sqrt{2}}{2} \end{bmatrix} \begin{Bmatrix} F_{3X} \\ F_{3Y} \end{Bmatrix} = \frac{\sqrt{2}}{2}(F_{3X} + F_{3Y}) = 0,$$

that is

$$F_{3X} + F_{3Y} = 0$$

Examples with Bar Elements

Applying the load and BC's in the structure FE equation by "deleting" the 1st, 2nd and 4th rows and columns, we have

$$1260\times10^5\begin{bmatrix}1 & -1 & 0\\ -1 & 1.5 & 0.5\\ 0 & 0.5 & 0.5\end{bmatrix}\begin{Bmatrix}u_2\\ u_3\\ v_3\end{Bmatrix}=\begin{Bmatrix}P\\ F_{3X}\\ F_{3Y}\end{Bmatrix}$$

Further, from the MPC and the force relation at node 3, the equation becomes

$$1260\times10^5\begin{bmatrix}1 & -1 & 0\\ -1 & 1.5 & 0.5\\ 0 & 0.5 & 0.5\end{bmatrix}\begin{Bmatrix}u_2\\ u_3\\ u_3\end{Bmatrix}=\begin{Bmatrix}P\\ F_{3X}\\ -F_{3X}\end{Bmatrix}$$

which is

$$1260\times10^5\begin{bmatrix}1 & -1\\ -1 & 2\\ 0 & 1\end{bmatrix}\begin{Bmatrix}u_2\\ u_3\end{Bmatrix}=\begin{Bmatrix}P\\ F_{3X}\\ -F_{3X}\end{Bmatrix}$$

Examples with Bar Elements

Solving this, we obtain the displacements

$$\begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = \frac{1}{2520\times 10^5}\begin{Bmatrix} 3P \\ P \end{Bmatrix} = \begin{Bmatrix} 0.01191 \\ 0.003968 \end{Bmatrix} \text{(m)}$$

From the global FE equation, we can calculate the reaction forces

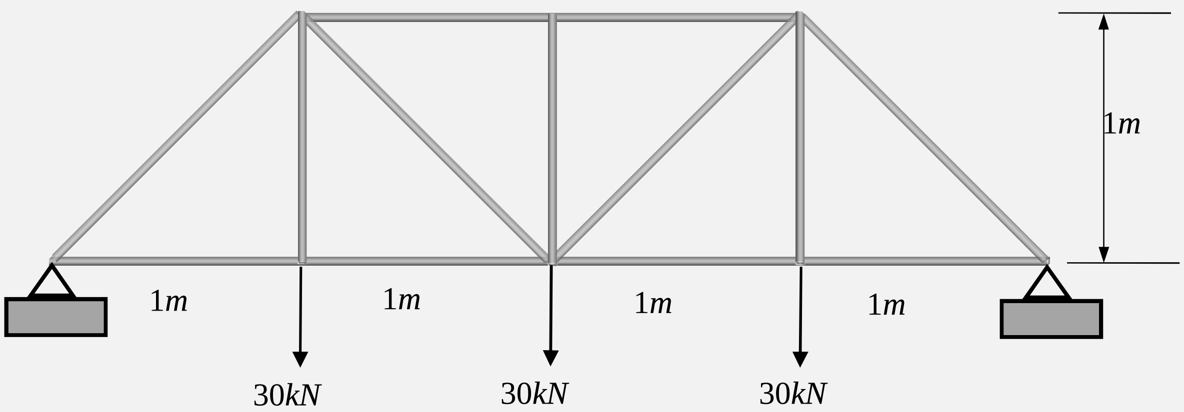
$$\begin{Bmatrix} F_{1X} \\ F_{1Y} \\ F_{2Y} \\ F_{3X} \\ F_{3Y} \end{Bmatrix} = 1260\times 10^5 \begin{bmatrix} 0 & -0.5 & -0.5 \\ 0 & -0.5 & -0.5 \\ 0 & 0 & 0 \\ -1 & 1.5 & 0.5 \\ 0 & 0.5 & 0.5 \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \\ v_3 \end{Bmatrix} = \begin{Bmatrix} -500 \\ -500 \\ 0.0 \\ -500 \\ 500 \end{Bmatrix} \text{(kN)}$$

Analysis of a truss bridge …

Case Study with *ANSYS Workbench*

Case Study with ANSYS Workbench

<Problem Description> Truss bridges can span long distances and support heavy weights without intermediate supports. They are economical to construct and are available in a wide variety of styles. Consider the following planar truss, constructed of wooden timbers, which can be used in parallel to form bridges. Determine the deflections at each joint of the truss under the given loading conditions.



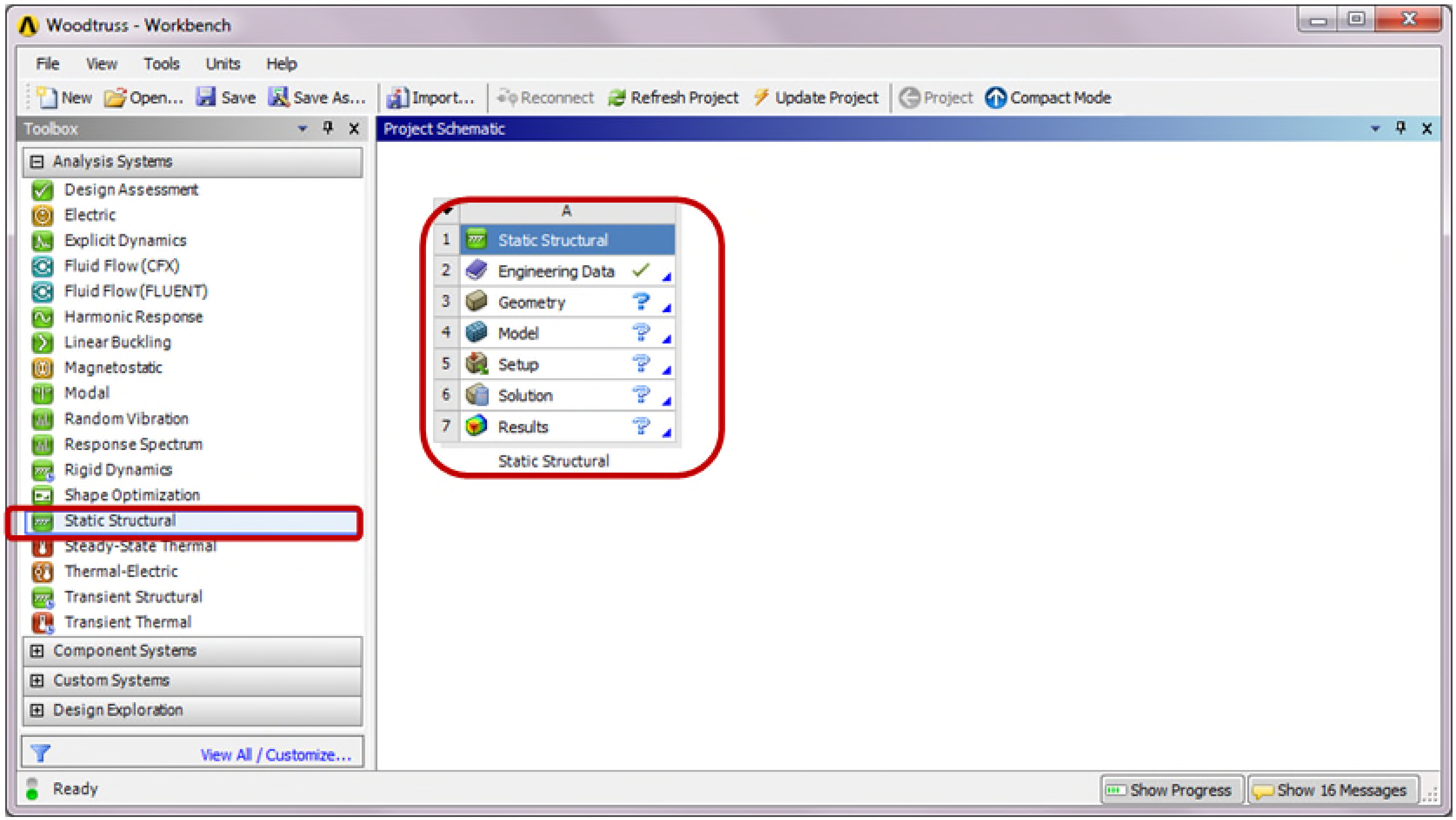
Material: Douglas Fir
$E = 13.1\ GPa$
$\nu = 0.29$
Member cross section:
height = 6 cm
width = 6 cm

Case Study with ANSYS Workbench

Step 1: Start an *ANSYS Workbench* Project

Launch *ANSYS Workbench* and save the blank project as ***'Woodtruss.wbpj'***.

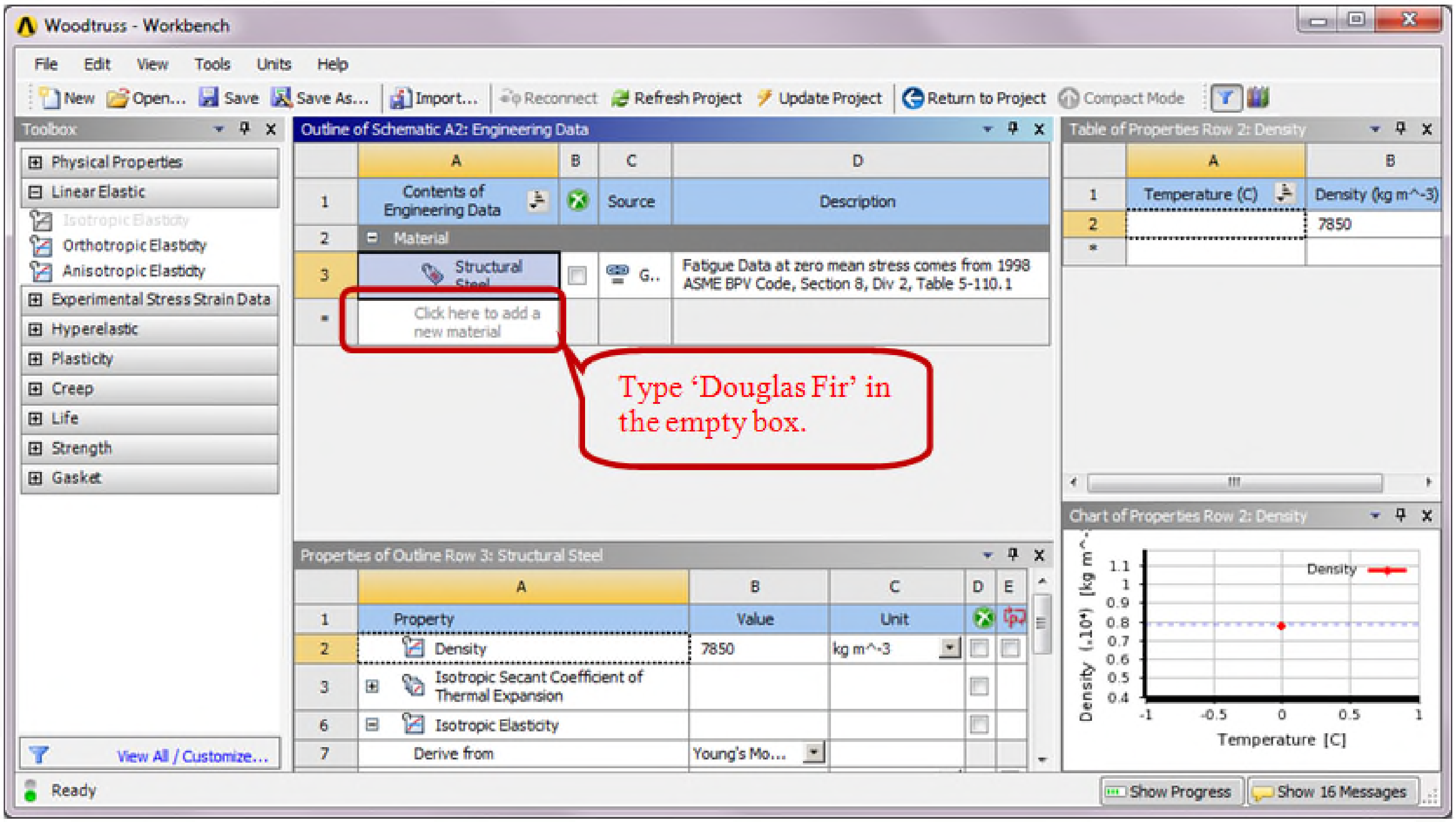
Step 2: Create a *Static Structural (ANSYS)* Analysis System



Case Study with ANSYS Workbench

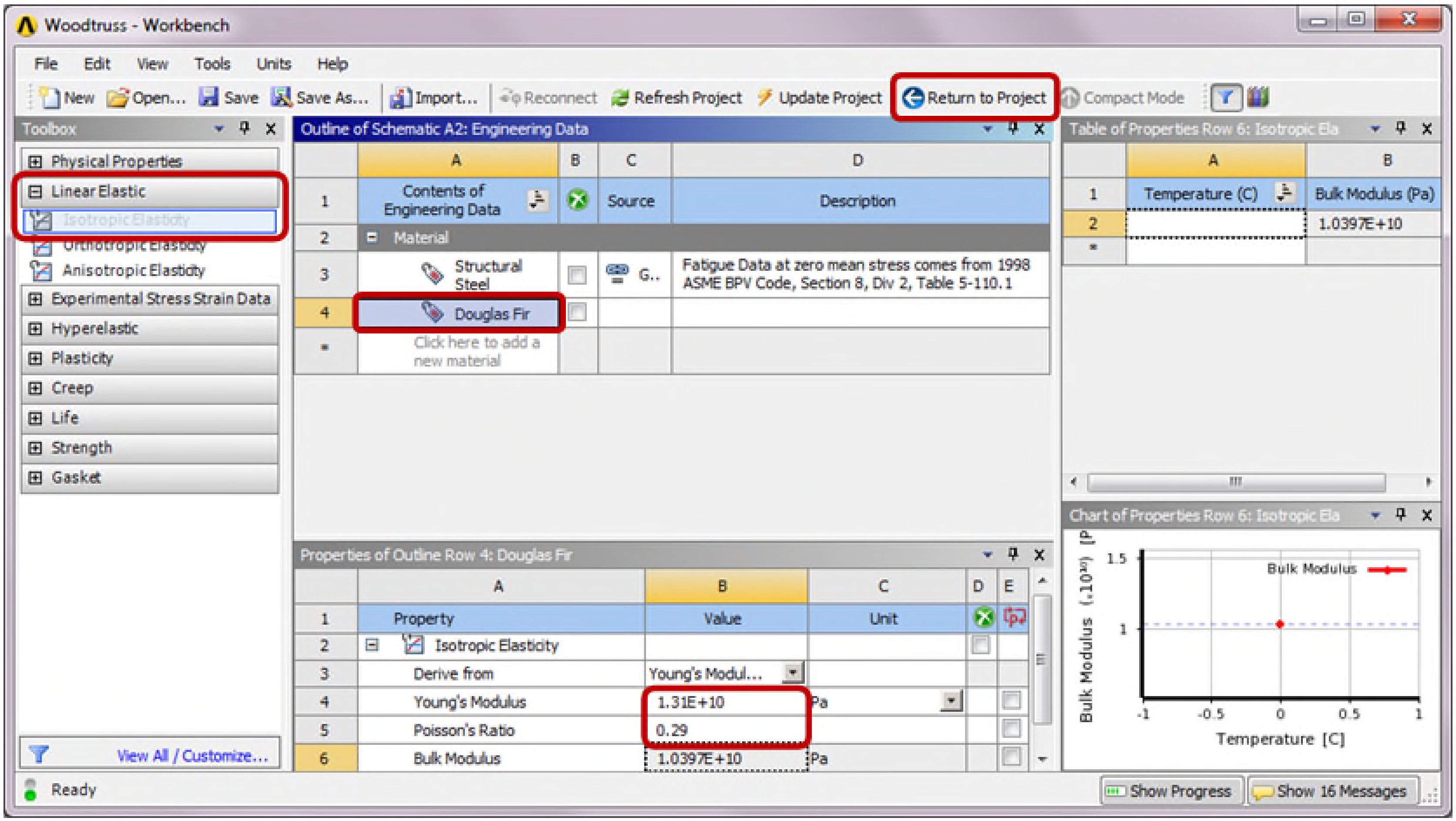
Step 3: Add a New Material

Double-click on the ***Engineering Data*** cell in the above ***Project Schematic*** to edit or add a material. In the following ***Engineering Data*** interface which replaces the ***Project Schematic***, click the empty box highlighted below and type a name, '***Douglas Fir***', for the new material.



Case Study with ANSYS Workbench

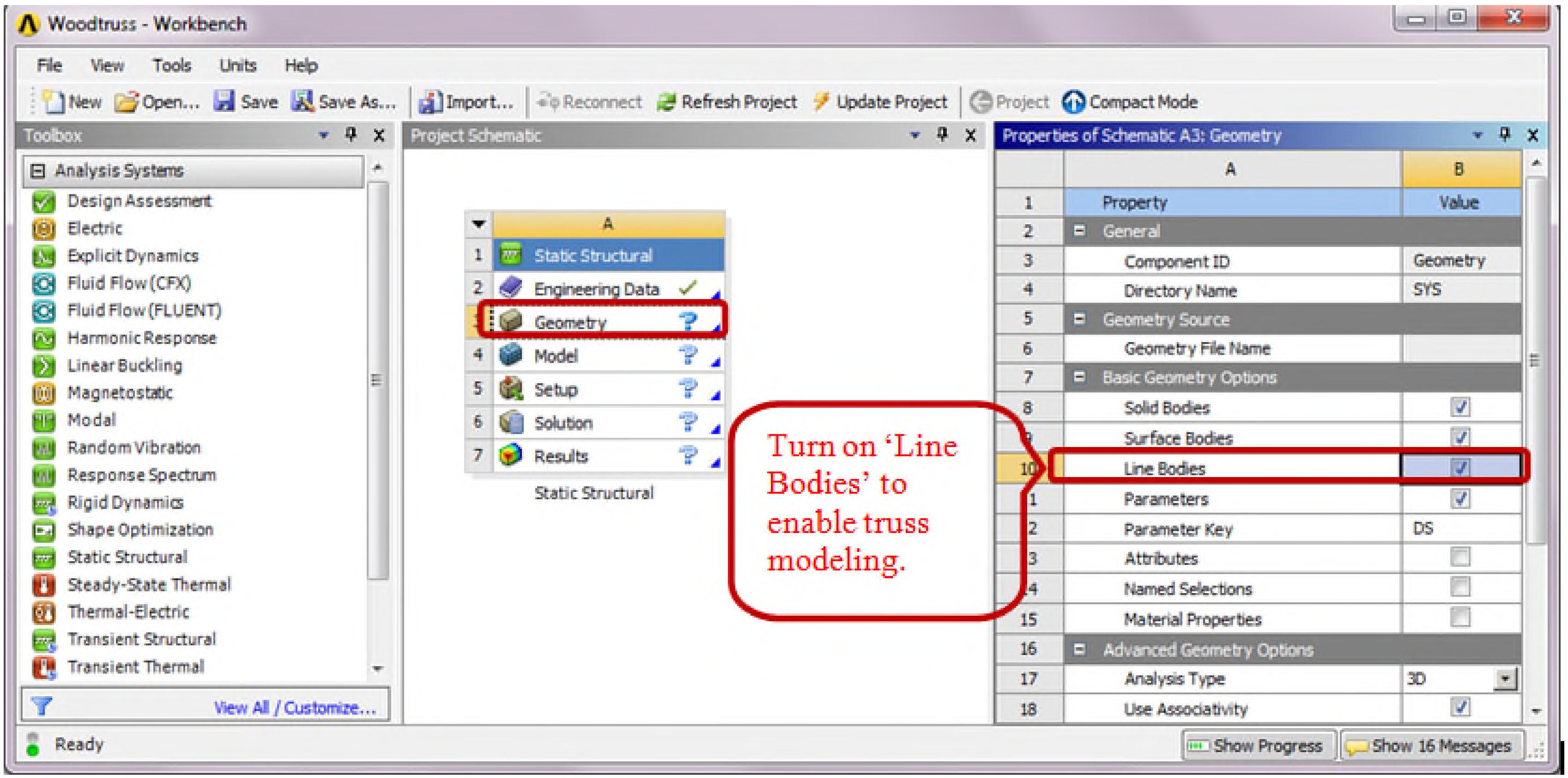
Select '***Douglas Fir***' from the ***Outline*** window, and double-click ***Isotropic Elasticity*** under ***Linear Elastic*** in the leftmost ***Toolbox*** window. Enter '***1.31E10***' for ***Young's Modulus*** and '***0.29***' for ***Poisson's Ratio*** in the bottom center ***Properties*** window.



Case Study with ANSYS Workbench

Step 4: <u>Launch the *DesignModeler* Program</u>

Ensure ***Line Bodies*** is checked in the ***Properties of Schematic A3: Geometry*** window. Double-click the ***Geometry*** cell to launch ***DesignModeler***, and select '***Meter***' as length unit in the ***Units*** pop-up window.

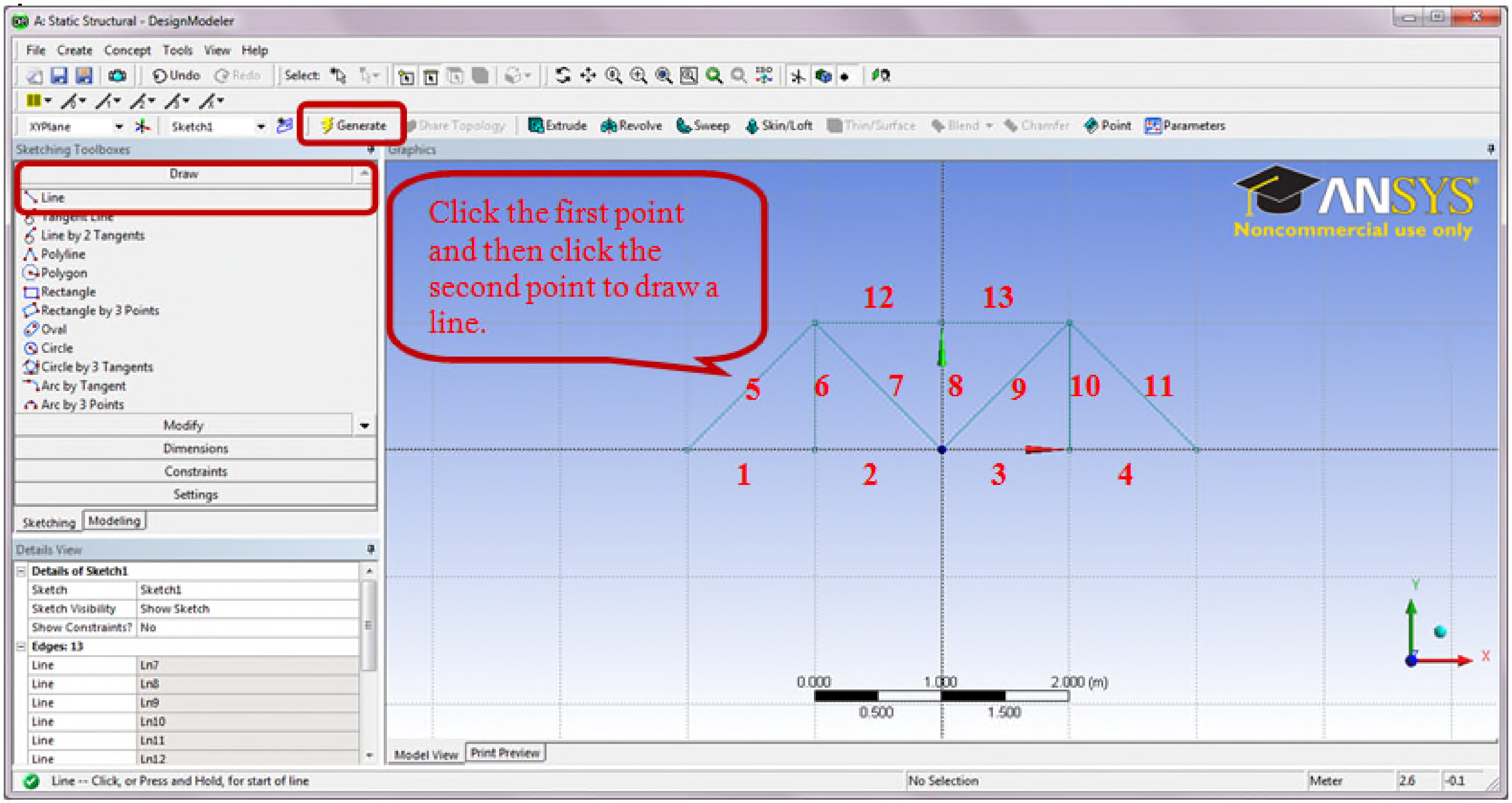


Case Study with ANSYS Workbench

Step 5: Create Line Sketch

Click the ***Sketching*** tab and select ***Settings***. Turn on ***Show in 2D*** and ***Snap*** under ***Grid*** options.

Draw 13 lines as shown in the sketch below. After completion, click ***Generate*** to create a line sketch.

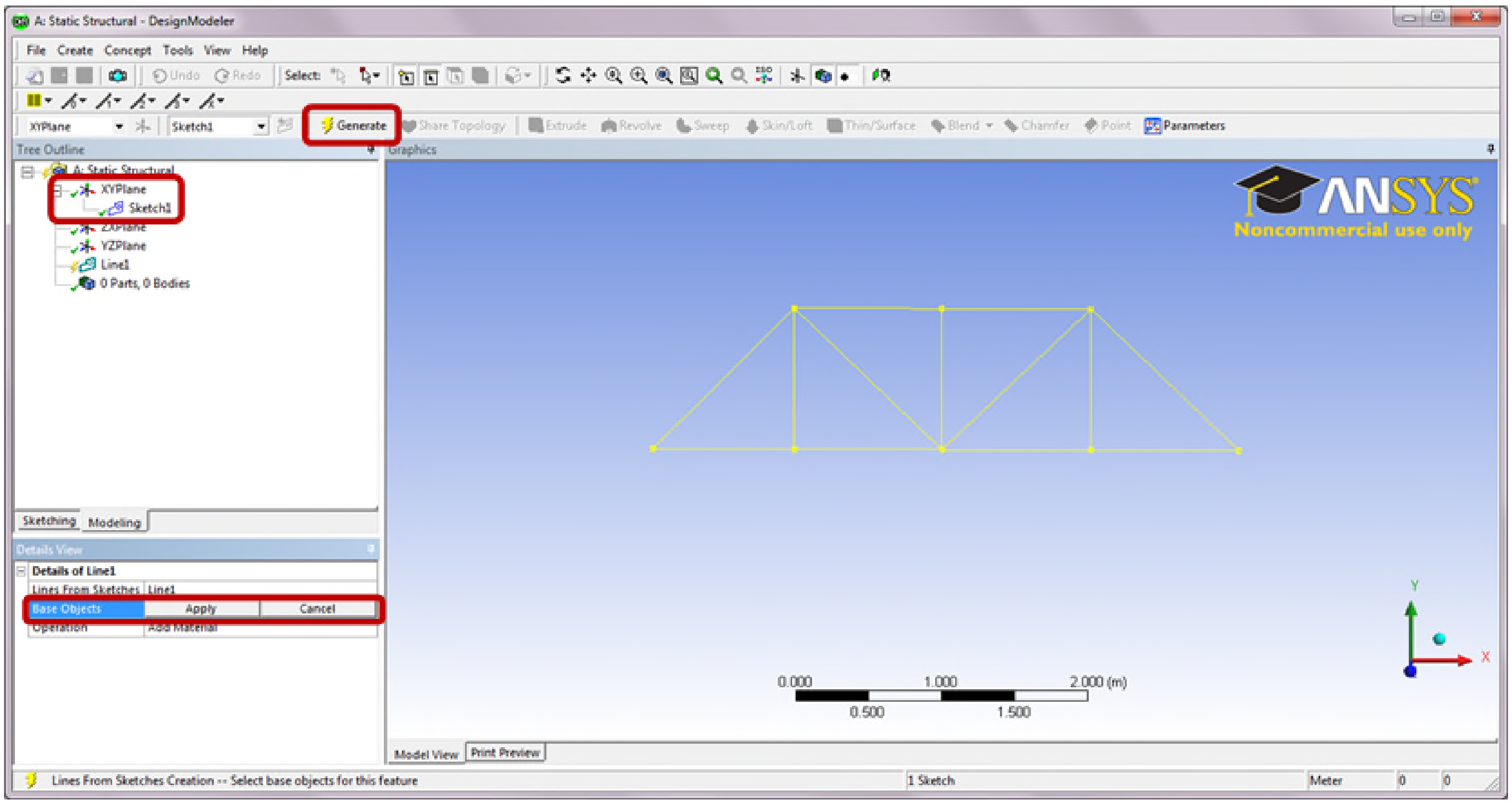


Case Study with ANSYS Workbench

Step 6: Create Line Body from Sketch

Check off the ***Grid*** options under ***Settings*** of ***Sketching Toolboxes***. Switch to the ***Modeling*** tab. Note that a new item named ***Sketch1*** now appears underneath ***XYPlane*** in the ***Tree Outline***.

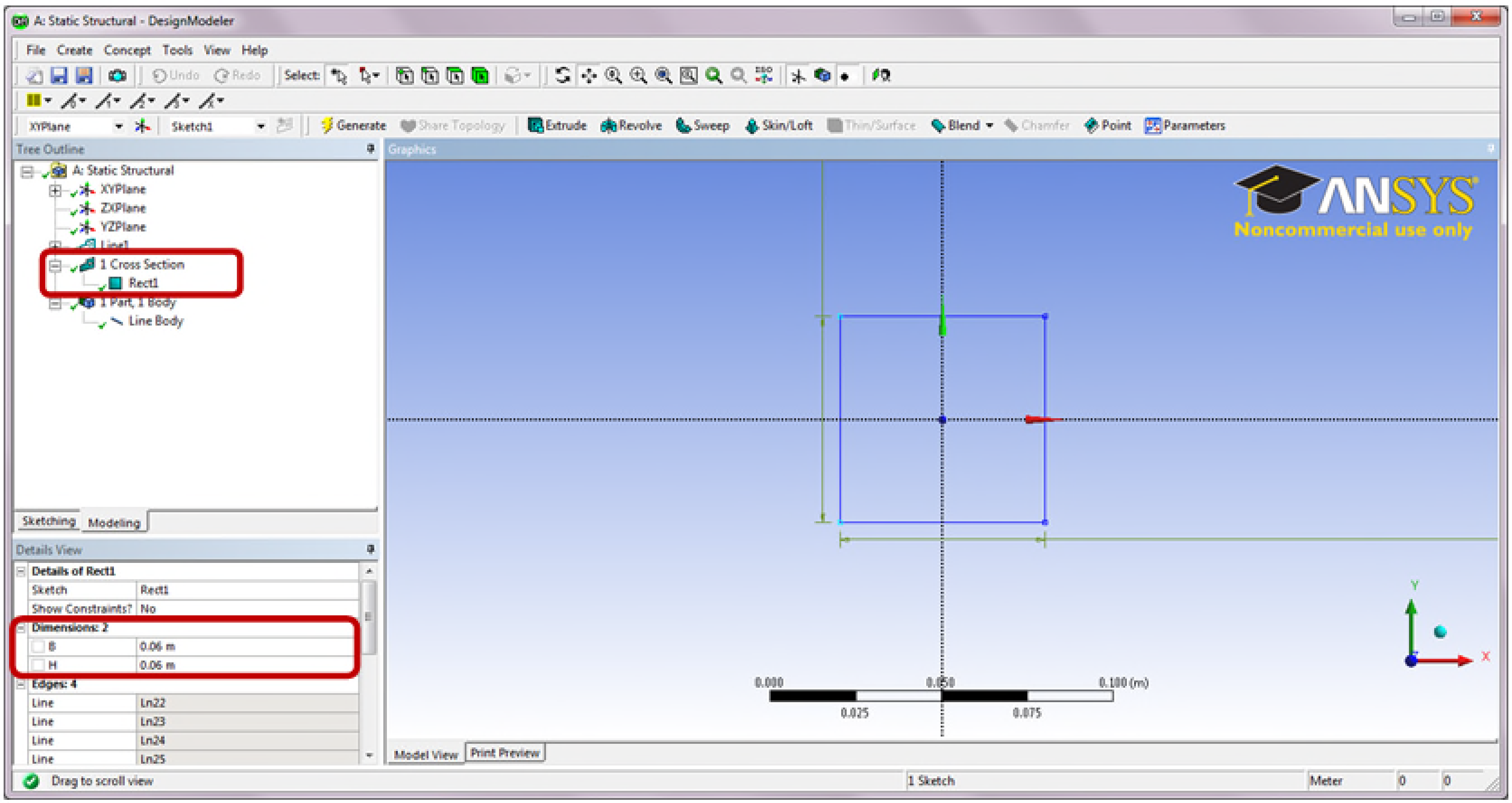
Select ***Sketch1*** from the ***Tree Outline*** and click ***Apply*** to confirm on the ***Base Objects*** selection in the ***Details of Line1***. Click ***Generate*** to complete the line body creation.



Case Study with ANSYS Workbench

Step 7: <u>Create a Cross Section</u>

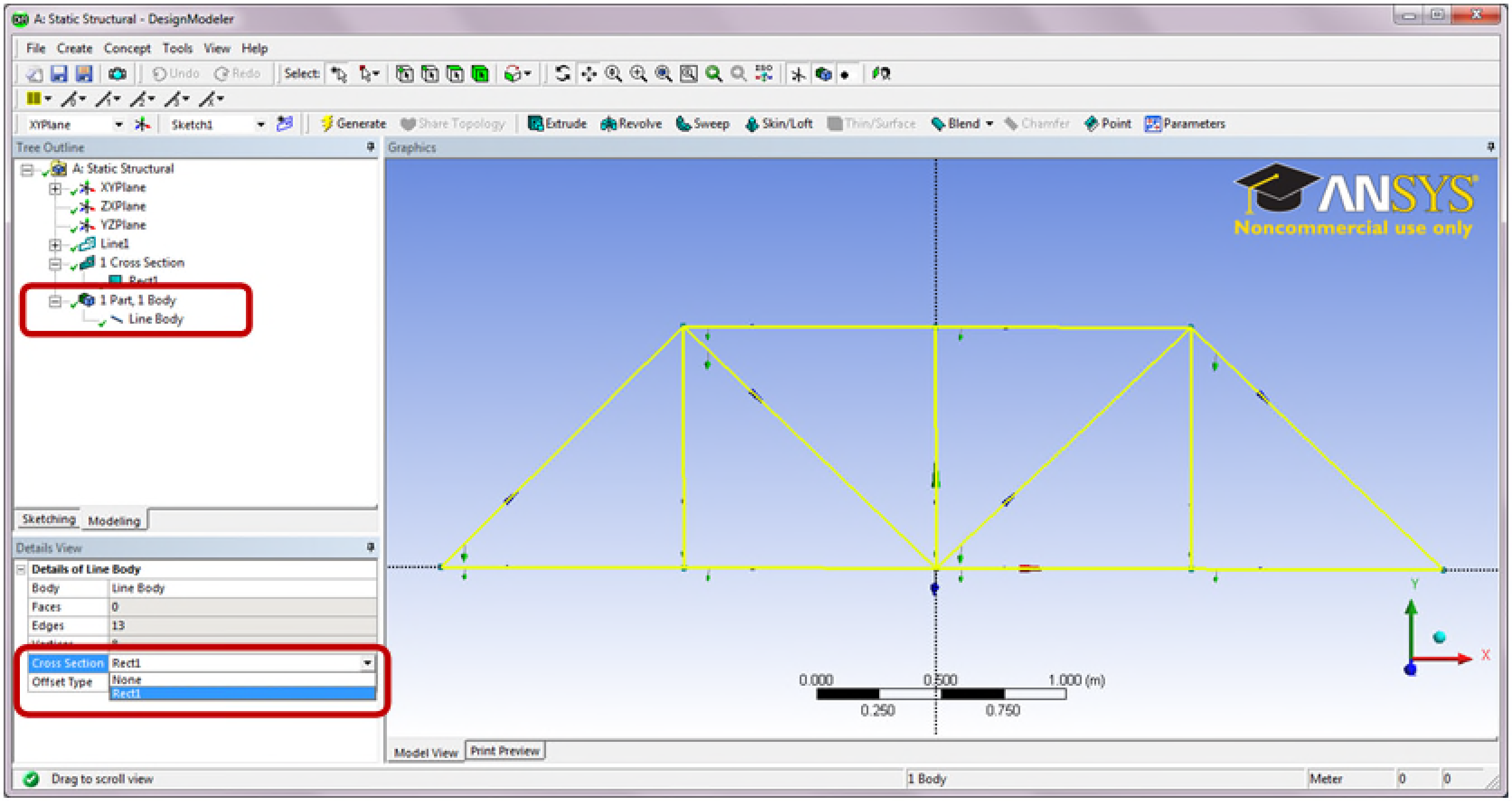
Select a ***Cross Section*** of ***Rectangular*** from the ***Concept*** drop-down menu. A new item named ***Rect1*** is now added underneath the ***Cross Section*** in the ***Tree Outline***. In the ***Details of Rect1*** under ***Dimensions***, enter '***0.06m***' for both ***B*** and ***H***.



Case Study with ANSYS Workbench

Step 8: <u>Assign Cross Section to Line Body</u>

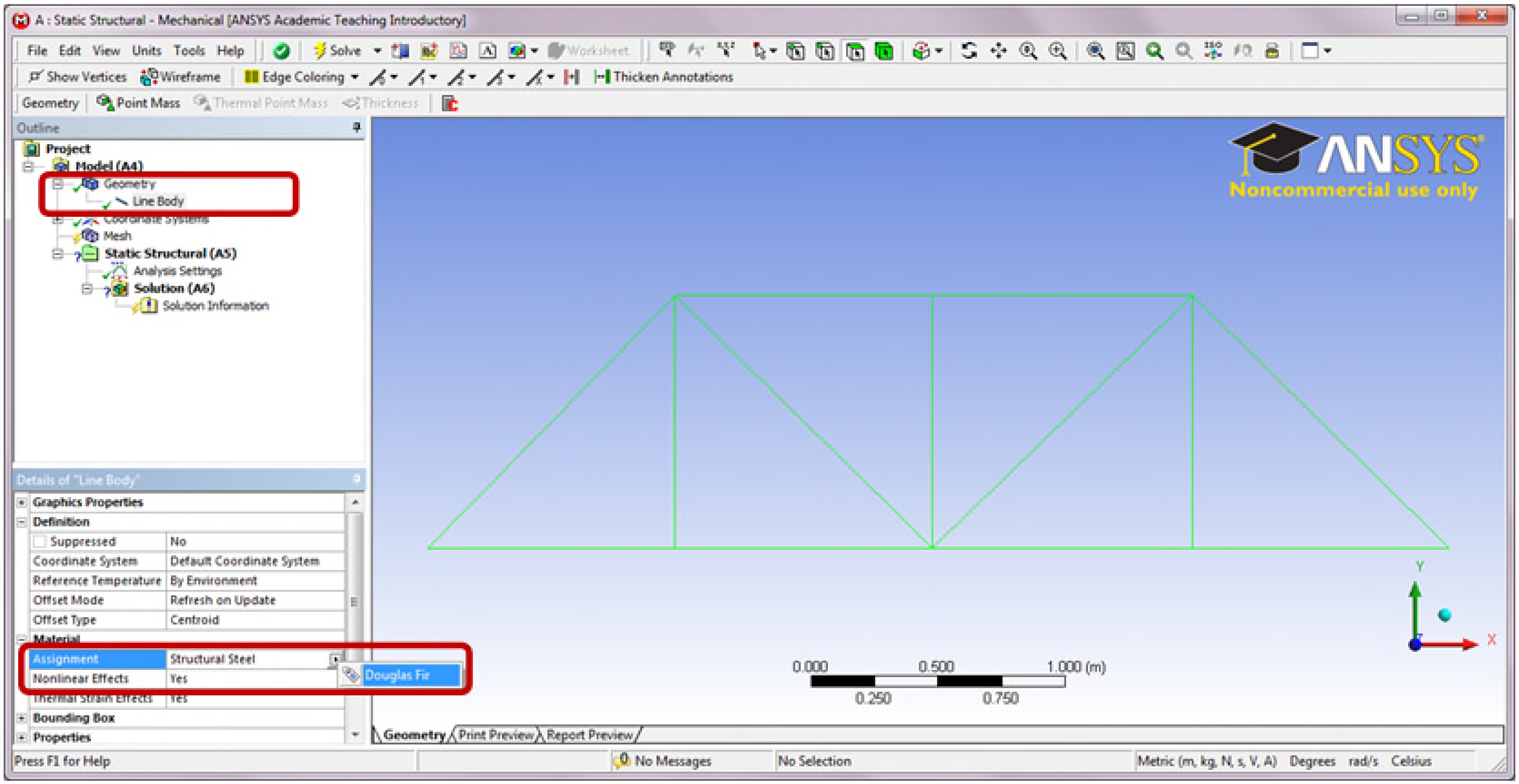
Select the ***Line Body*** underneath ***1Part, 1 Body*** in the ***Tree Outline***. In the ***Details of Line Body***, assign ***Rect1*** to the ***Cross Section*** selection. Click ***Close DesignModeler*** to exit the program.



Case Study with ANSYS Workbench

Step 9: <u>Launch the *Static Structural (ANSYS)* Program</u>

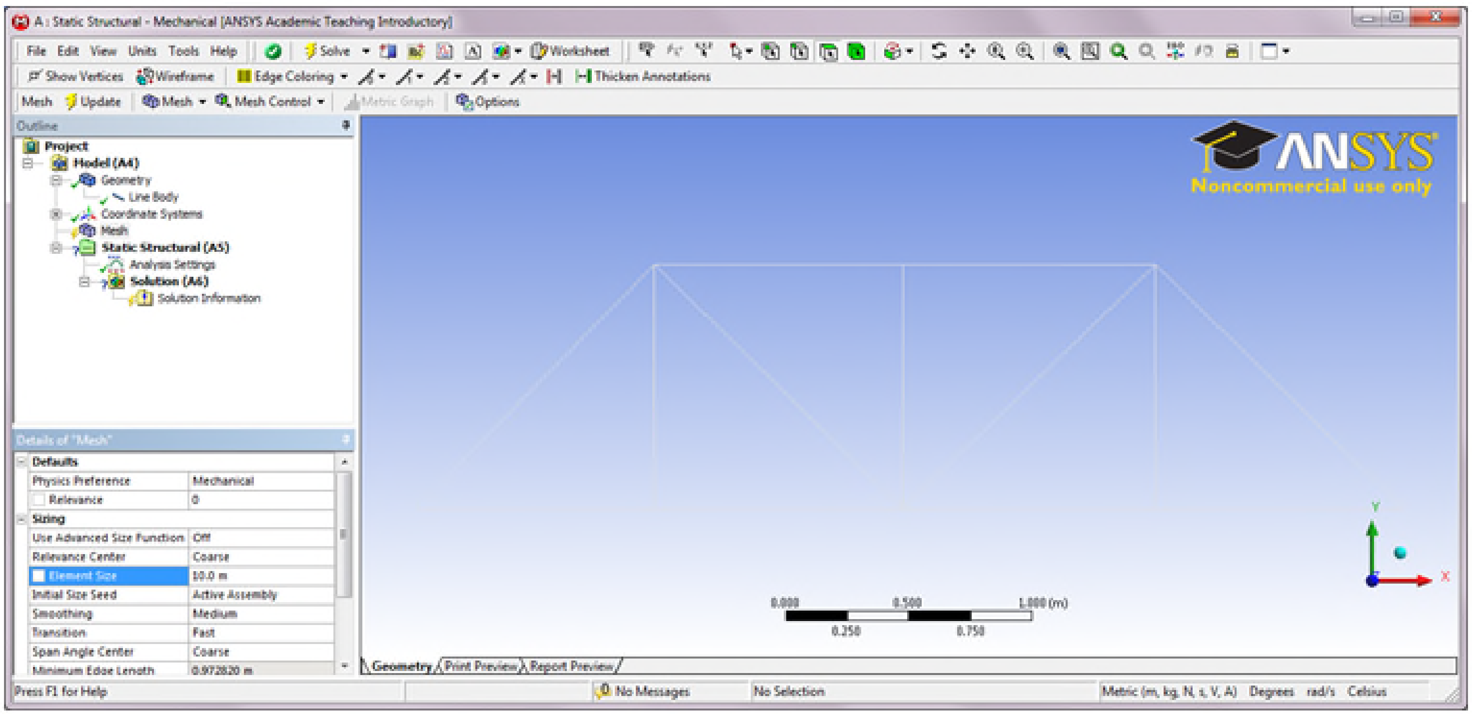
Double-click the ***Model*** cell to launch the ***Static Structural (ANSYS)*** program. Note that in the ***Details of "Line Body"*** the material is assigned to ***Structural Steel*** by default. Click to the right of the ***Assignment*** field and select ***Douglas Fir*** from the drop-down context menu.



Case Study with ANSYS Workbench

Step 10: <u>Generate Mesh</u>

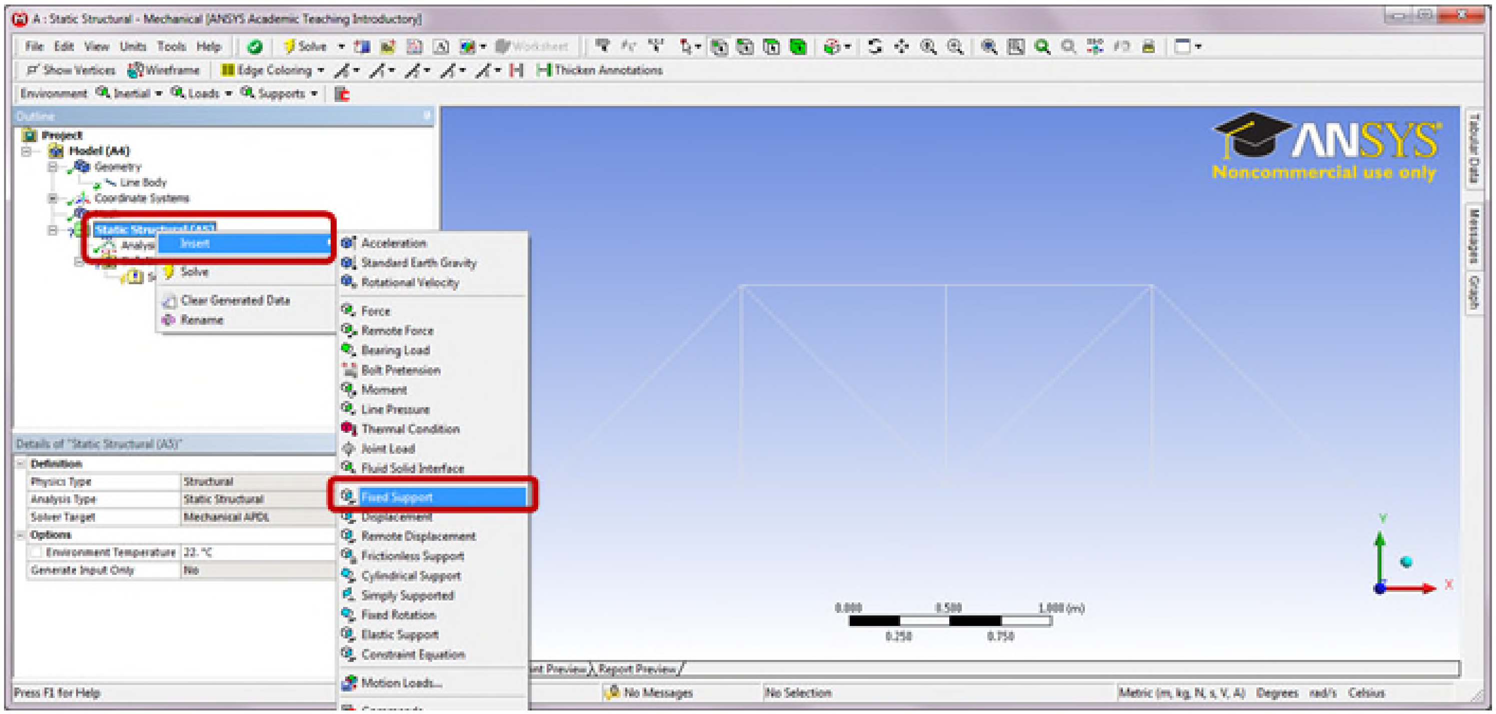
In the ***Details of "Mesh"***, enter a fairly large number, say,'***10m***', for the ***Element Size***, to ensure each member is meshed with only one element. In the ***Outline*** of ***Project***, right-click on ***Mesh*** and select ***Generate Mesh***.



Case Study with ANSYS Workbench

Step 11: <u>Apply Boundary Conditions</u>

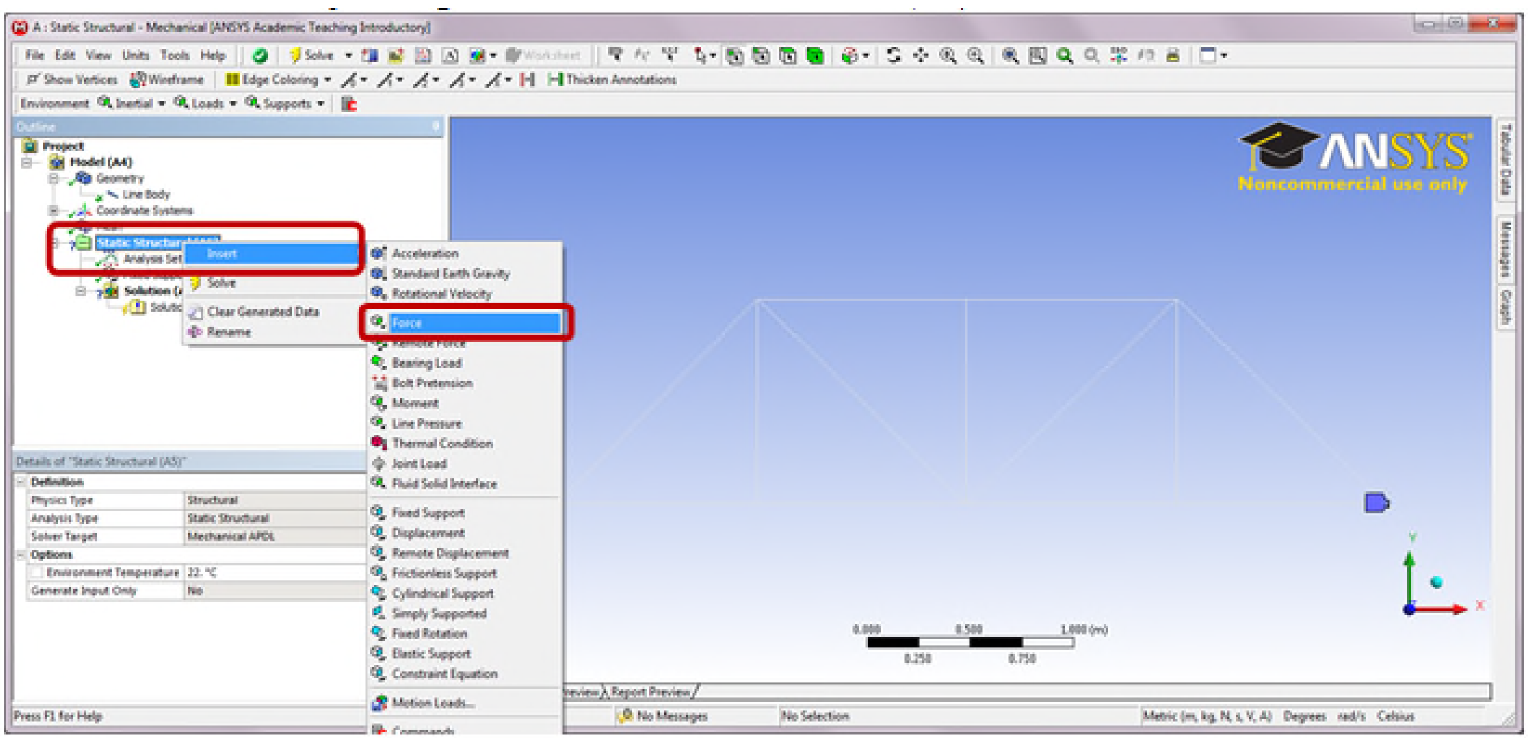
In the ***Outline*** of ***Project***, right-click on ***Static Structural (A5)*** and select ***Insert*** and then ***Fixed Support***. After completion, a ***Fixed Support*** item is added underneath ***Static Structural (A5)*** in the project outline tree.



Case Study with ANSYS Workbench

Step 12: **<u>Apply Loads</u>**

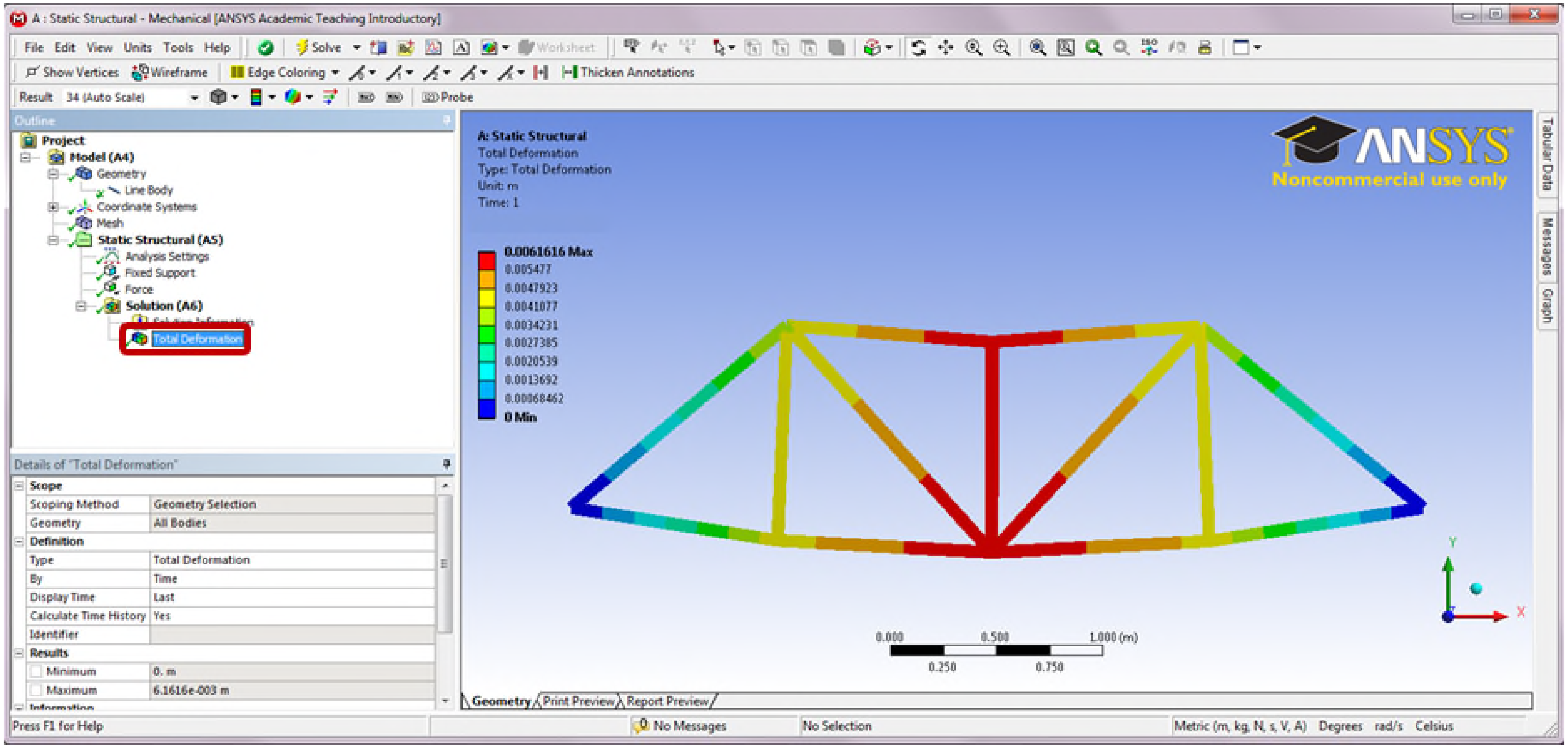
In the ***Outline*** of ***Project***, right-click on ***Static Structural (A5)*** and select ***Insert*** and then ***Force***. In the ***Details of "Force"***, click ***Apply*** to confirm on the Geometry selection. Also underneath the ***Details***, change the ***Define By*** selection to ***Components*** and enter ***'-90000N'*** for the ***Y Component***.



Case Study with ANSYS Workbench

Step 13: <u>Retrieve Solution</u>

Insert a ***Total Deformation*** item by right-clicking on ***Solution (A6)*** in the ***Outline*** tree. Right-click on ***Solution (A6)*** in the ***Outline*** tree and select ***Solve***. The program will start to solve the model. After completion, click ***Total Deformation*** in the ***Outline*** to review the total deformation results.



Summary

In this chapter,

- ❑ We study the bar elements which can be used in truss analysis.
- ❑ The concept of the shape functions is introduced and the derivations of the stiffness matrices using the energy approach are introduced.
- ❑ Treatment of distributed loads is discussed and several examples are studied.
- ❑ A planar truss structure is analyzed using *ANSYS Workbench*. It provides basic modeling techniques and shows step-by-step how *Workbench* can be used to determine the deformation and reaction forces in trusses.