# Course in finite element simulations using Abaqus

## Topic 1b introduction:
## The Concept of Finite Element Analysis (FEA)

**Esmaeil Tohidlou**

**Department of Materials Engineering**

**University of Sistan and Baluchestan**

**etohidlou@gmail.com**

**2016**

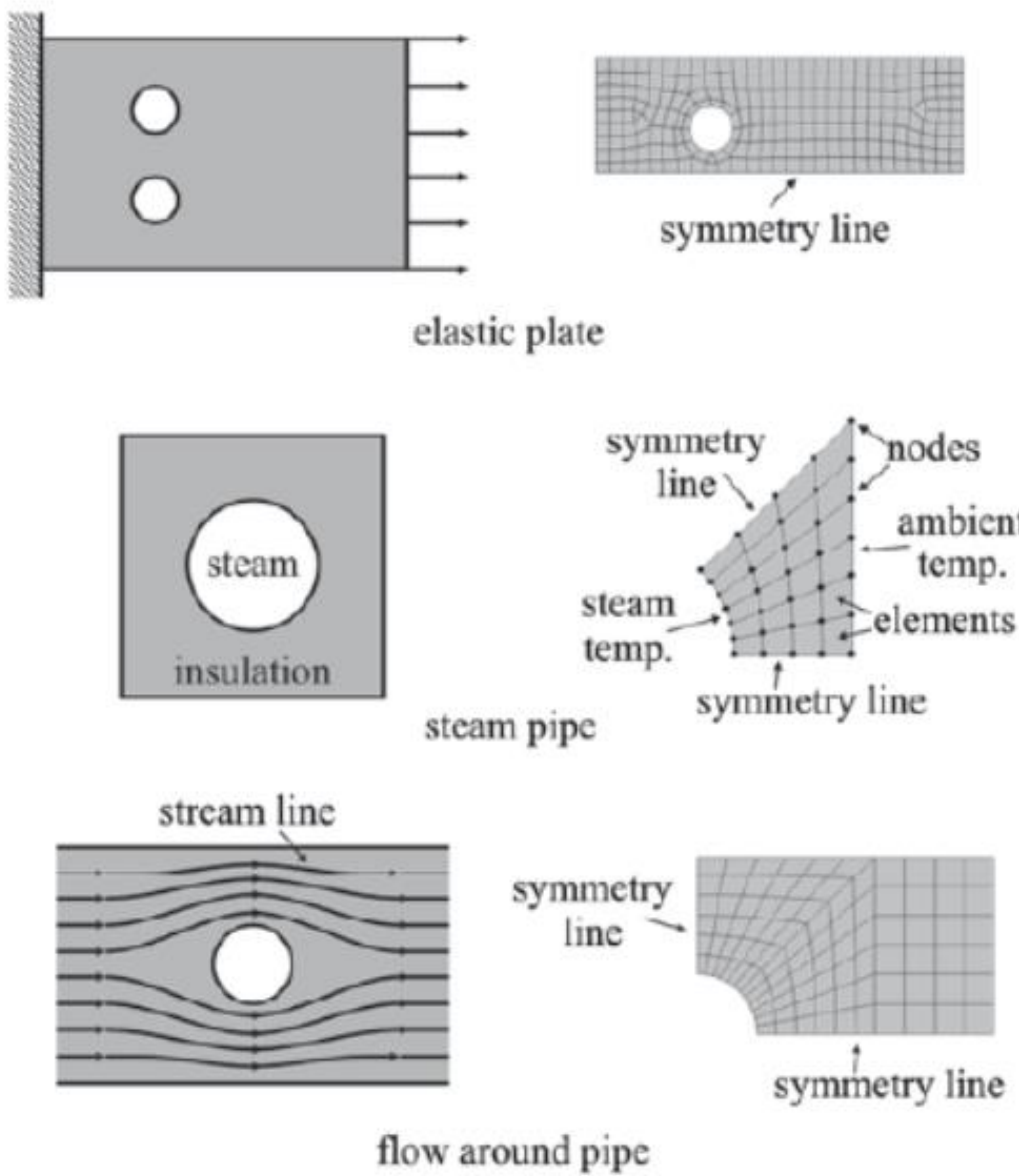
## Concept of FEA:

The basis of FEA relies on the decomposition of the domain into a finite number of subdomains (Elements) for which the systematic approximate solution is constructed by applying the variational or weighted residual methods.

The FEM requires the following major steps:

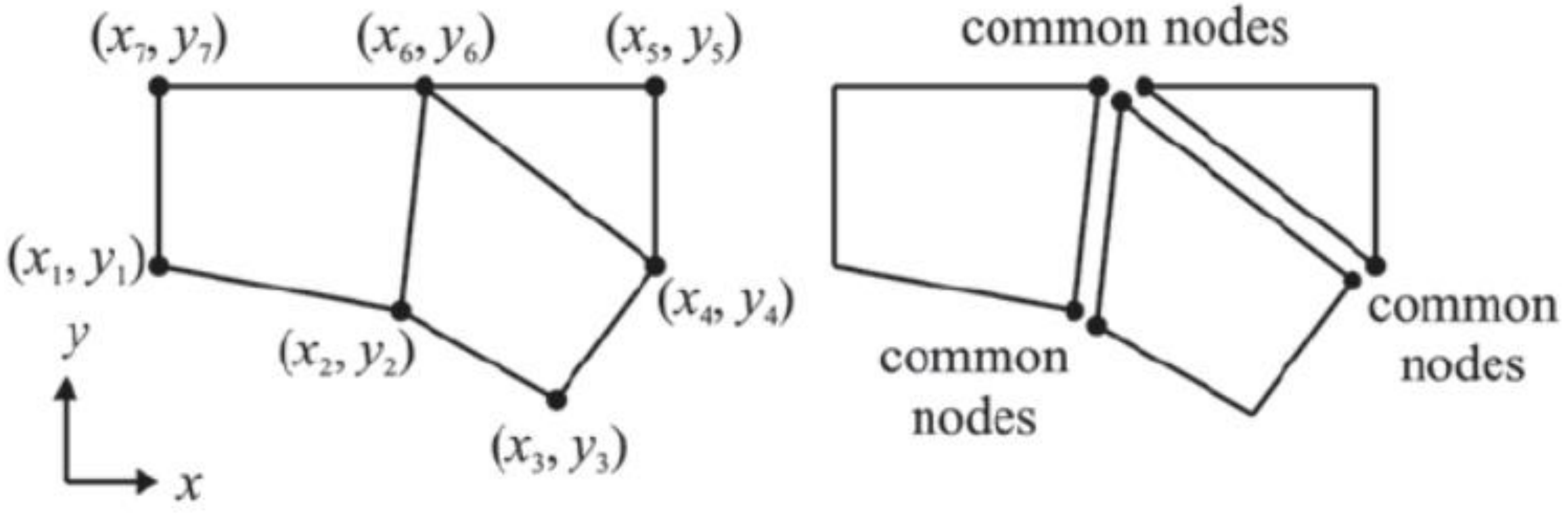
- Discretization of the domain into finite number of sundomains (elements)
- Selection of interpolation functions
- Development of element matrix for the subdomain (element)
- Asselbly of the element matrices for each subdomain to obtain global matrix for the entire domain
- Imposition of the boundary conditions
- Solution of equations $\mathbf{Ku} = \mathbf{F}$
- Additional computations (If desired)

The FEA representation of practical engineering problems



## Node

The transformation of the practical engineering problem to a mathematical representation is achieved by discretizing the domain of interest in to elements (elements). These elements are connected to each other by their nodes. A node specifies the coordinate location in space where degree of freedom and actions of the physical problem exist. The nodal unknown(s) in the matrix system of equations represents one (or more) of the primary field variable. Nodal variables assigned to an element are called dgrees of freedom of the element.

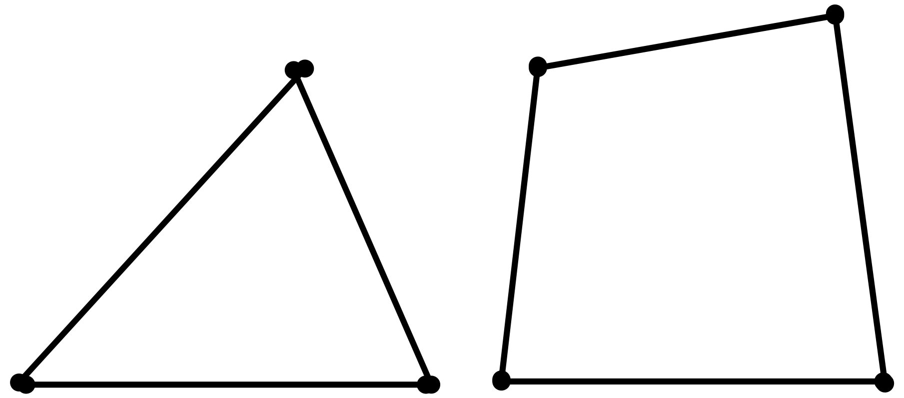


## Element

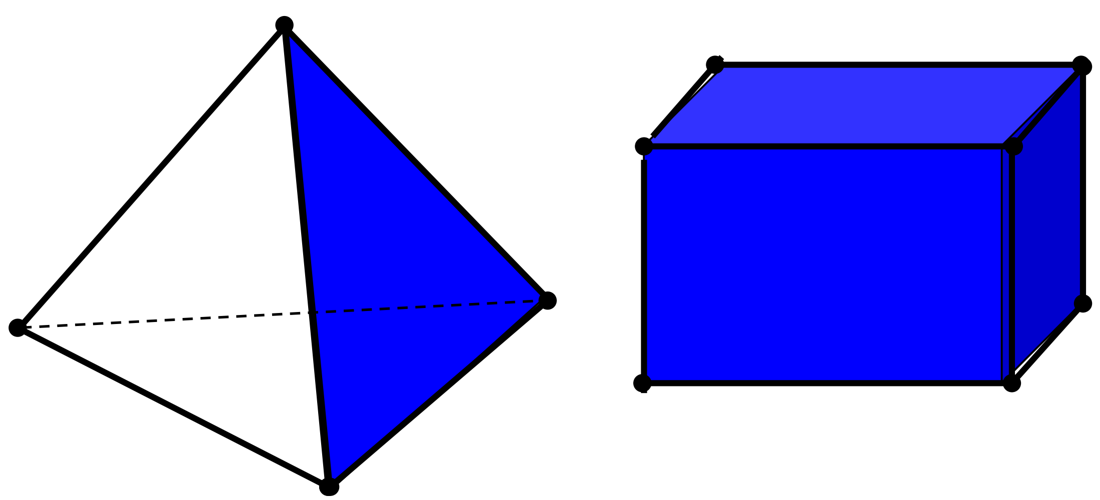
Depending on the geometry and the physical nature of the problem, the domain of the interest can be discretized by employing line, area, or volume elements. Some of the common elements in the FEM are shown in Figure.

**One-Dimensional Elements**
**Line**
**Rods, Beams, Trusses, Frames**

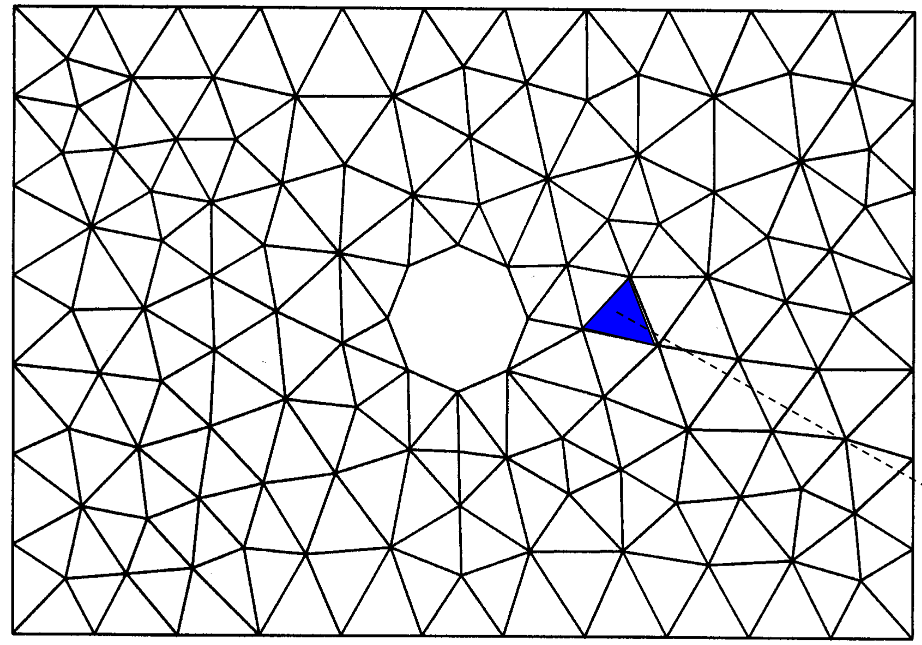
**Two-Dimensional Elements**
**Triangular, Quadrilateral**
**Plates, Shells, 2-D Continua**



**Three-Dimensional Elements**
**Tetrahedral, Rectangular Prism (Brick)**
**3-D Continua**

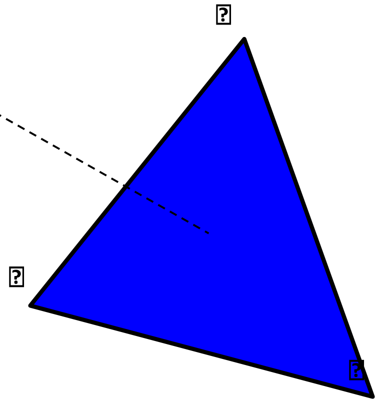


# Two-Dimensional Discretization Refinement

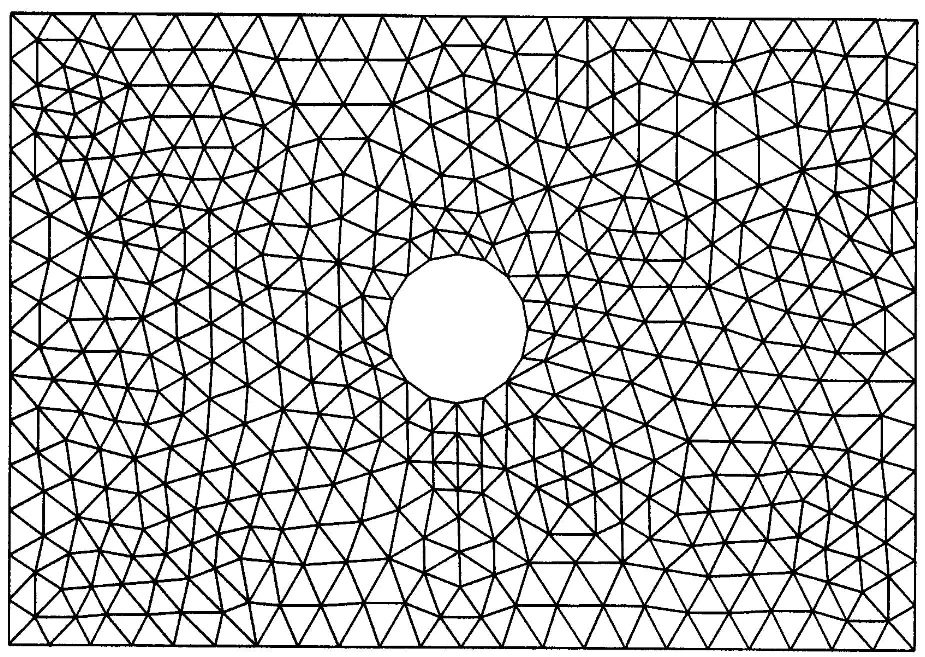


(Discretization with 228 Elements)

(Node)



(Triangular Element)



(Discretization with 912 Elements)

## Development of Finite Element Equation

- The Finite Element Equation Must Incorporate the Appropriate Physics of the Problem
- For Problems in Structural Solid Mechanics, the Appropriate Physics Comes from Either Strength of Materials or Theory of Elasticity
- FEM Equations are Commonly Developed Using *Direct*, *Variational-Virtual Work* or *Weighted Residual* Methods

### Direct Method

Based on physical reasoning and limited to cases, this method is worth studying because it enhances physical understanding of the process

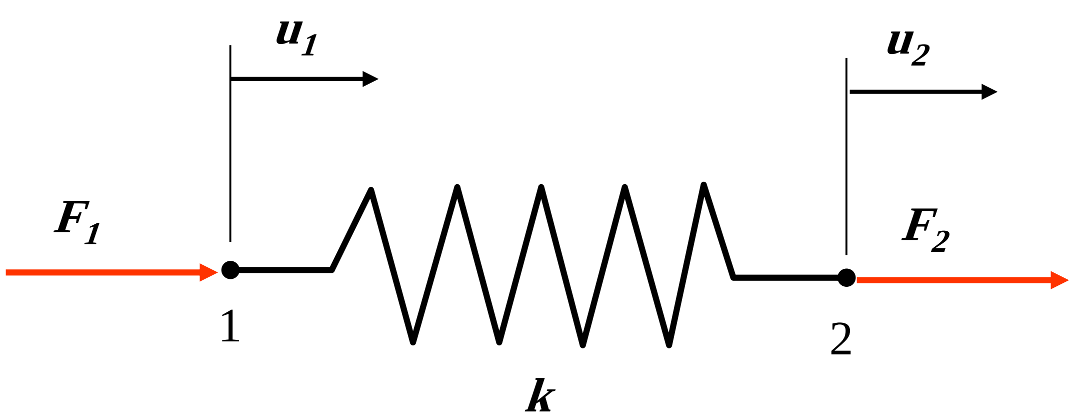
### Variational-Virtual Work Method

Based on the concept of virtual displacements, leads to relations between internal and external virtual work and to minimization of system potential energy for equilibrium

### Weighted Residual Method

Starting with the governing differential equation, special mathematical operations develop the "weak form" that can be incorporated into a FEM equation. This method is particularly suited for problems that have no variational statement. For stress analysis problems, a Ritz-Galerkin WRM will yield a result identical to that found by variational methods.

# Formulation of the stiffness matrix: Direct method




$$\text{Equilibrium at Node 1} \Rightarrow F_1 = ku_1 - ku_2$$

$$\text{Equilibrium at Node 2} \Rightarrow F_2 = -ku_1 + ku_2$$

or in Matrix Form

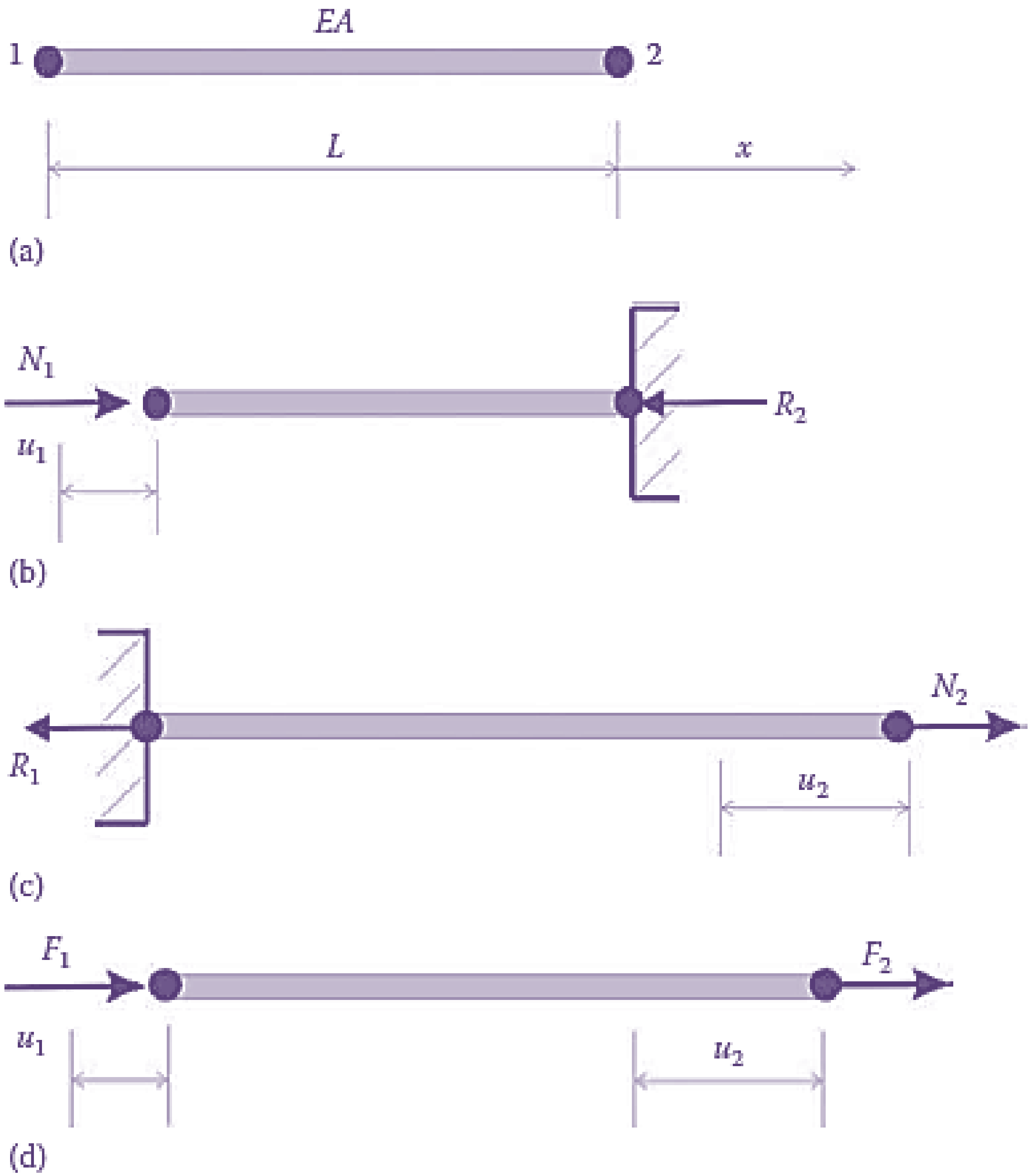
$$\begin{bmatrix} k & -k \\ -k & k \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \end{Bmatrix}$$

Stiffness Matrix

Nodal Force Vector

$$[K]\{u\} = \{F\}$$

# Bar Element




$$\mathbf{N}_1 = \frac{AE}{L}\mathbf{u}_1, \quad \mathbf{R}_2 = -\frac{AE}{L}\mathbf{u}_1$$

$$\mathbf{N}_2 = \frac{AE}{L}\mathbf{u}_2, \quad \mathbf{R}_1 = -\frac{AE}{L}\mathbf{u}_2$$

$$\mathbf{F}_1 = \frac{AE}{L}\mathbf{u}_1 - \frac{AE}{L}\mathbf{u}_2$$

$$\mathbf{F}_2 = \frac{AE}{L}\mathbf{u}_2 - \frac{AE}{L}\mathbf{u}_1$$

$$\mathbf{K}_e\mathbf{u}_e = \mathbf{F}_e$$

Bar element:
a) geometry,
b) nodal force applied at node 1,
c) nodal force applied at node 2,
d) nodal forces at both nodes

# Two-Dimensional Truss Element


Degrees of freedom of a rod element in a two-dimensional space

The nodal degrees of freedom (nodal displacement) of the rod element becom four, as represented in the Figure

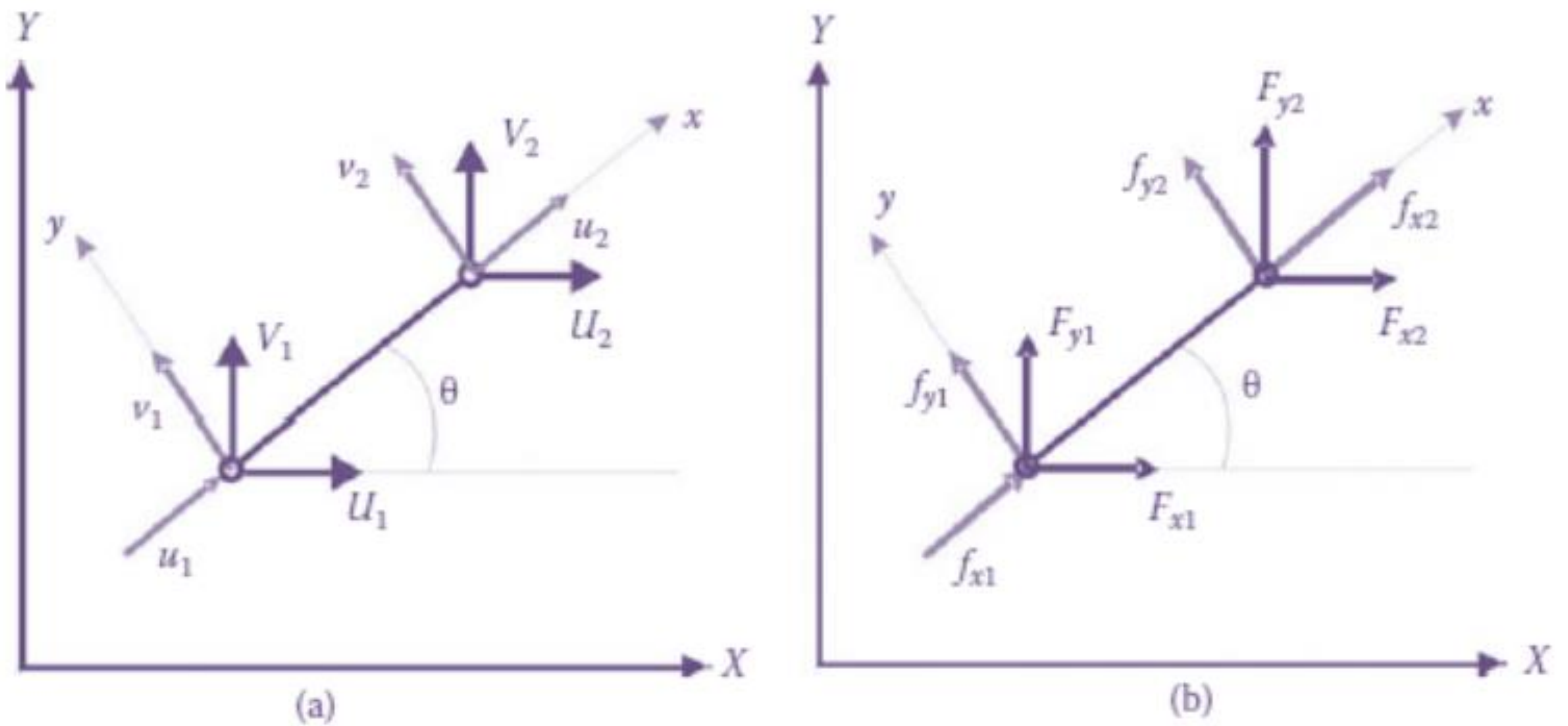
$$\{d_e\} = \{\mathbf{u}_1, \mathbf{v}_1, \mathbf{u}_2, \mathbf{v}_2\}^T$$

The corresponding stifness matrix becomes:

$$[K_e] = \begin{bmatrix} AE/L & 0 & -AE/L & 0 \\ 0 & 0 & 0 & 0 \\ -AE/L & 0 & AE/L & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix}$$

The second and fourth columns and rows associated with transversal displacements are null since the truss member has axial deformation only.

Another problem arises from the fact that all truss members do not have the same orientation is that when it comes to assemble the global stiffness , we need to have the element degrees of freedom (nodal displacements) given in terms of the common reference axes of the truss.




Truss element oriented at an arbitrary angle: a) Nodal displacement, b) Nodal forces

Figure shows two sets of nodal displacements; the first set $(\boldsymbol{u}, \boldsymbol{v})$ is given in terms of the local set of axis $(x, y)$ associated with the element, while the second set of displacements $(\boldsymbol{U}, \boldsymbol{V})$ is associated with the global set of axis $(X, Y)$.
The element stifness matrix is expressed in the terms of the local displacements $\boldsymbol{u}$ and $\boldsymbol{v}$**.** In order to assemble with the stiffness matrices of the other elements to form the global stiffness matrix of the whole system, it should be transformed such that it is expressed in terms of the global displacement $\boldsymbol{U}$ and $\mathbf{V}$**.**

If we consider node 1, it can be seen that the displacements $\boldsymbol{U}_1$ and $\boldsymbol{V}_1$ can be written in terms of $\boldsymbol{u}_1$ and $\boldsymbol{v}_1$ as

$$\boldsymbol{U}_1 = \boldsymbol{u}_1 cos\theta - \boldsymbol{v}_1 sin\theta$$
$$\boldsymbol{V}_1 = \boldsymbol{u}_1 sin\theta + \boldsymbol{v}_1 cos\theta$$

In a similar fashion, $\boldsymbol{U}_2$ and $\boldsymbol{V}_2$ can be expressed in terms of $\boldsymbol{u}_2$ and $\boldsymbol{v}_2$ as

$$\boldsymbol{U}_2 = \boldsymbol{u}_2 cos\theta - \boldsymbol{v}_2 sin\theta$$
$$\boldsymbol{V}_2 = \boldsymbol{u}_2 sin\theta + \boldsymbol{v}_2 cos\theta$$

From the above equations

$$\begin{Bmatrix} U_1 \\ V_1 \\ U_2 \\ V_2 \end{Bmatrix} = \begin{bmatrix} \cos\theta & -\sin\theta & 0 & 0 \\ \sin\theta & \cos\theta & 0 & 0 \\ 0 & 0 & \cos\theta & -\sin\theta \\ 0 & 0 & \sin\theta & \cos\theta \end{bmatrix} \begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \end{Bmatrix}$$

Or in a more compact form as

$$\{\bar{\mathbf{d}}_e\} = [\mathbf{C}]\{\mathbf{d}_e\}$$

The matrix $[\mathbf{C}]$ is called transformation matrix. It is an orthogonal matrix with determinant equal to one. Its inverse is simply equal to its transpose: $[\mathbf{C}]^{-1} = [\mathbf{C}]^T$

The vector of the global nodal forces $\{\overline{f_e}\} = \{F_{x1}, F_{y1}, F_{x2}, F_{y2}\}^T$ may be also obtained from the vector of local nodal forces $\{f_e\} = \{f_{x1}, f_{y1}, f_{x2}, f_{y2}\}^T$ as

$$\{\overline{f_e}\} = [C]\{f_e\}$$

In the local coordinate system, the force–displacement relation is given as

$$[K_e]\{d_e\} = \{f_e\}$$

Using $\{d_e\} = [C]^T\{\overline{d_e}\}$ and $\{f_e\} = [C]^T\{\overline{f_e}\}$, and substitutin yields

$$[K_e][C]^T\{\overline{d_e}\} = [C]^T\{\overline{f_e}\}$$

Premultiplying both sides by $[C]$ yields

$$[C][K_e][C]^T\{\overline{d_e}\} = \{\overline{f_e}\}$$

which can be rewritten as

$$[\overline{K_e}]\{\overline{d_e}\} = \{\overline{f_e}\}$$

with

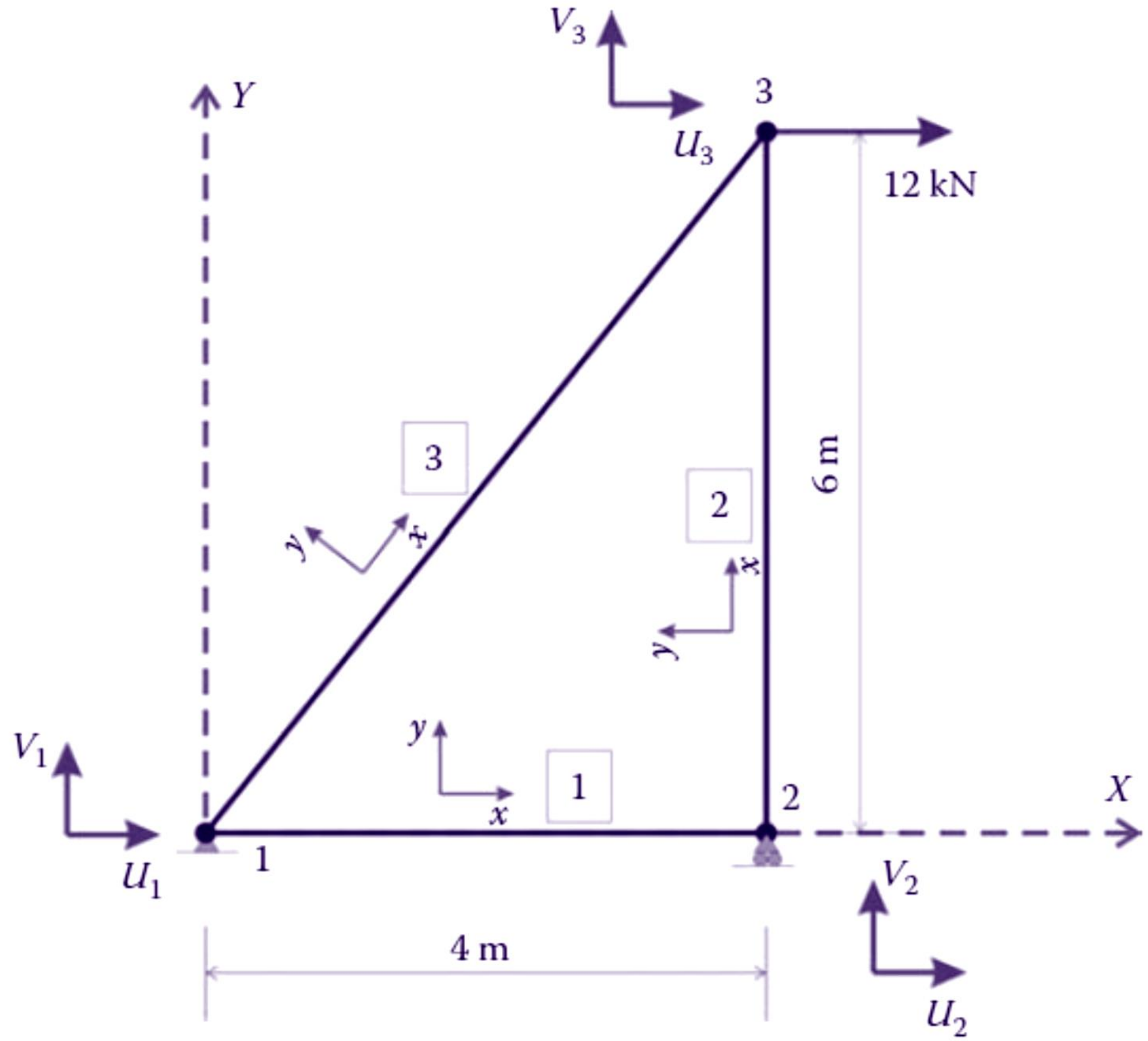
$$[\overline{K_e}] = [C][K_e][C]^T$$

The matrix $[\overline{K_e}]$ is called the element stiffness matrix in the global coordinate system; it relates the global nodal displacements to the global nodal forces.

# Global stiffness matrix assembly

To illustrate how element's stiffness matrices are put together to form the global stiffness matrix, we proceed with a very simple example. First, we number all elements and the nodes as well as identifying the nodal degrees of freedom.

Material Properties:
elastic modulus(E)=200 Gpa
cross-sectional area(A)= 2300 $mm^2$




Model of a truss structure

## Elements' stiffness matrix in local coordinates

The stifness matrix for element 1 in its local coordinate:

$$[K_1]_L = \begin{bmatrix} 115000 & 0 & -115000 & 0 \\ 0 & 0 & 0 & 0 \\ -115000 & 0 & 115000 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix}$$

The stifness matrix for element 2 in its local coordinate:

$$[K_2]_L = \begin{bmatrix} 76666.67 & 0 & -76666.67 & 0 \\ 0 & 0 & 0 & 0 \\ -76666.67 & 0 & 76666.67 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix}$$

The stifness matrix for element 3 in its local coordinate:

$$[K_3]_L = \begin{bmatrix} 63791.43 & 0 & -63791.43 & 0 \\ 0 & 0 & 0 & 0 \\ -63791.43 & 0 & 63791.43 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix}$$

## Elements' stiffness matrices in Global coordinates

In order to assemble elements' stiffness matrices, they need to be transform from their local coorinate systems $(x, y)$ to the global coordinate system $(X, Y)$.

For element 1 the transformation matrix $[C_1]$ is given as

$$[C_1] = \begin{bmatrix} \cos(0) & -\sin(0) & 0 & 0 \\ \sin(0) & \cos(0) & 0 & 0 \\ 0 & 0 & \cos(0) & -\sin(0) \\ 0 & 0 & \sin(0) & \cos(0) \end{bmatrix} = \begin{bmatrix} 1 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 \\ 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & 1 \end{bmatrix}$$

The stiffness matrix of elemnt 1 in the global coordinates system remains unchanged, so

$$[K_1]_G = [C_1][K_1]_L[C_1]^T = \begin{bmatrix} 115000 & 0 & -115000 & 0 \\ 0 & 0 & 0 & 0 \\ -115000 & 115000 & 0 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix}$$

For element 2 the transformation matrix $[C_2]$ and the stifness matrix in the global coordinate $[K_2]_G$ are

$$[C_2] = \begin{bmatrix} 0 & -1 & 0 & 0 \\ 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & -1 \\ 0 & 0 & 1 & 0 \end{bmatrix} ; \ [K_2]_G = \begin{bmatrix} 0 & 0 & 0 & 0 \\ 76666.67 & 0 & -76666.67 & 0 \\ 0 & 0 & 0 & 0 \\ -76666.67 & 0 & 76666.67 & 0 \end{bmatrix}$$

For element 3 the transformation matrix $[C_3]$ and the stifness matrix in the global coordinate $[K_3]_G$ are

$$[C_3] = \begin{bmatrix} 0.554699 & -0.832051 & 0 & 0 \\ 0,832051 & 0.554699 & 0 & 0 \\ 0 & 0 & 0.554699 & -0.832051 \\ 0 & 0 & 0,832051 & 0.554699 \end{bmatrix}$$

$$[K_3]_G = \begin{bmatrix} 19628 & 29442 & -19628 & -29442 \\ 29442 & 44163 & -29442 & -44163 \\ -19628 & -29442 & 19628 & 29442 \\ -29442 & -44163 & 29442 & 44163 \end{bmatrix}$$

## Global matrix assebly

The above mentioned truss has six degrees of freedom $\{\mathbf{U_1}, \mathbf{V_1}, \mathbf{U_2}, \mathbf{V_2}, \mathbf{U_3}, \mathbf{V_3}\}$, that is two degrees of freedom per node. So, the stiffness matrix must therefore have six lines and six columns each corresponding to a degree of freedom:

$$[\mathbf{K}] = \begin{array}{c} \\ U_1 \\ V_1 \\ U_2 \\ V_2 \\ U_3 \\ V_3 \end{array} \begin{array}{c} \begin{array}{cccccc} U_1 & V_1 & U_2 & V_2 & U_3 & V_3 \end{array} \\ \begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix} \end{array}$$

To populate the global stiffness matrix, imagine three hypothesis states:

1)Only element 1 is present

2)Only element 2 is present

3)Only element 3 is present

For element 1

$$[\mathbf{K}] = \begin{array}{c} \\ U_1 \\ V_1 \\ U_2 \\ V_2 \\ U_3 \\ V_3 \end{array} \begin{array}{c} \begin{array}{cccccc} U_1 & V_1 & U_2 & V_2 & U_3 & V_3 \end{array} \\ \begin{bmatrix} 115000 & 0 & -115000 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ -115000 & 0 & 115000 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix} \end{array}$$

For element 2

$$[\mathbf{K}] = \begin{array}{c} \\ U_1 \\ V_1 \\ U_2 \\ V_2 \\ U_3 \\ V_3 \end{array} \begin{array}{c} \begin{array}{cccccc} U_1 & V_1 & U_2 & V_2 & U_3 & V_3 \end{array} \\ \begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 76666.67 & 0 & -76666.67 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & -76666.67 & 0 & 76666.67 \end{bmatrix} \end{array}$$

For element 3

$$[\mathbf{K}] = \begin{array}{c} \\ U_1 \\ V_1 \\ U_2 \\ V_2 \\ U_3 \\ V_3 \end{array} \begin{array}{c} \begin{array}{cccccc} U_1 & V_1 & U_2 & V_2 & U_3 & V_3 \end{array} \\ \begin{bmatrix} 19628 & 29442 & 0 & 0 & -19628 & -29442 \\ 29442 & 44163 & 0 & 0 & -29442 & -44163 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ -19628 & -29442 & 0 & 0 & 19628 & 29442 \\ -29442 & -44163 & 0 & 0 & 29442 & 44163 \end{bmatrix} \end{array}$$

By direct addition of the preceding matrices, the global structure stiffness matrix is obtained as

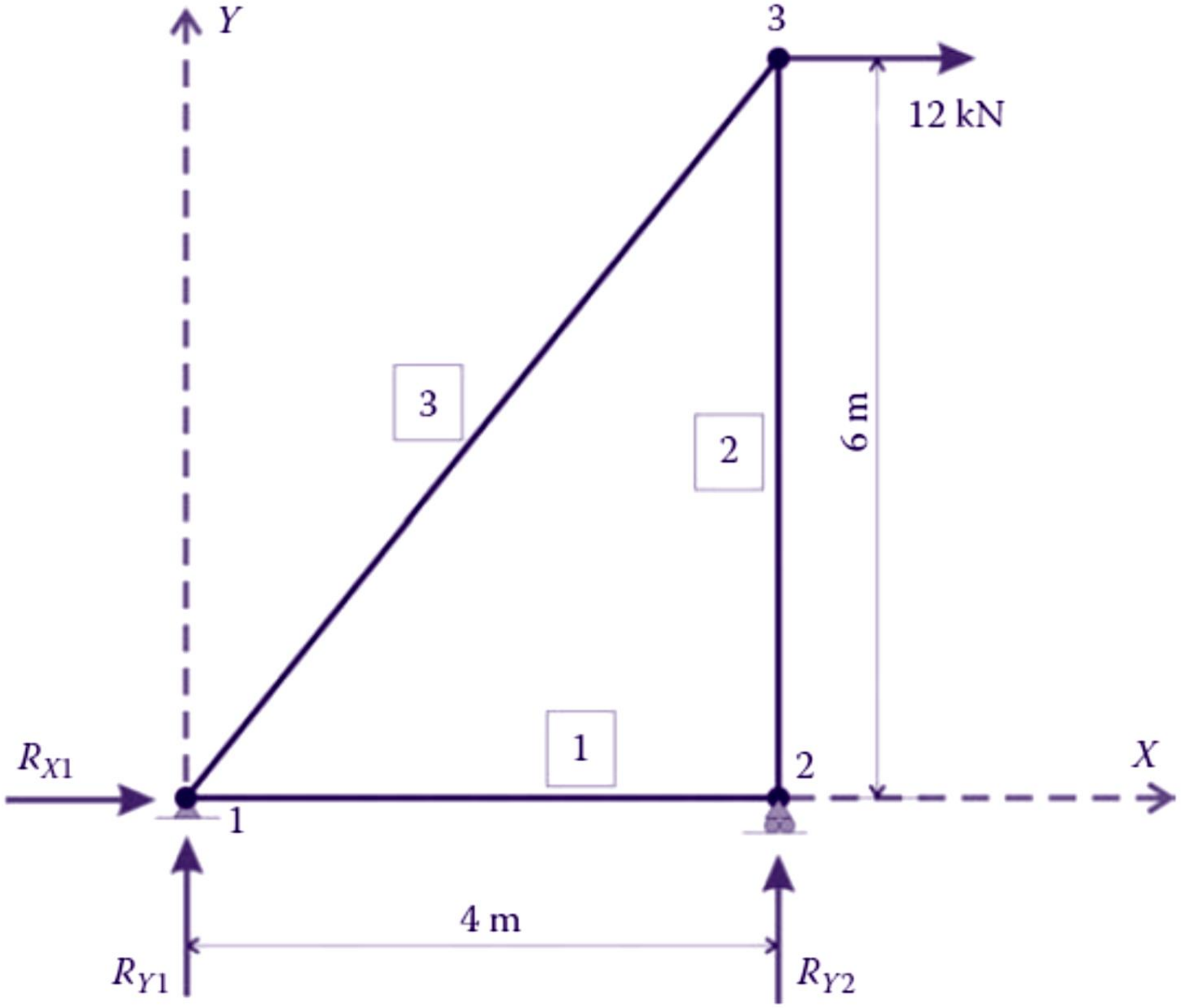
$$[\mathbf{K}] = \begin{array}{c} \\ U_1 \\ V_1 \\ U_2 \\ V_2 \\ U_3 \\ V_3 \end{array} \begin{array}{c} \begin{array}{cccccc} U_1 & V_1 & U_2 & V_2 & U_3 & V_3 \end{array} \\ \begin{bmatrix} 115000+19628 & 29442 & -115000 & 0 & -19628 & -29442 \\ 29442 & 44163 & 0 & 0 & -29442 & -44163 \\ -115000 & 0 & 115000 & 0 & 0 & 0 \\ 0 & 0 & 0 & 76666.67 & 0 & -76666.67 \\ -19628 & -29442 & 0 & 0 & 19628 & 29442 \\ -29442 & -44163 & 0 & -76666.67 & 29442 & 44163+76666.67 \end{bmatrix} \end{array}$$

$$[\mathbf{K}] = \sum_{e=1}^{N} \mathbf{k}_e$$

## Global force vector assembly

Figure shows a free body diagram where all the external forces acting on the truss are presented.

$$\{\mathbf{F}\} = \begin{Bmatrix} \mathbf{R}_{x_1} \\ \mathbf{R}_{y_1} \\ \mathbf{0} \\ \mathbf{R}_{y_2} \\ 12000 \\ \mathbf{0} \end{Bmatrix}$$




Free body diagram of the truss

## Boundary conditions

Once the global stiffnes matrix and global force vector are assembeld, the equilibrium equations of the truss are written as

$$\begin{bmatrix} 134628 & 29442 & -115000 & 0 & -19628 & -29442 \\ 29442 & 44163 & 0 & 0 & -29442 & -44163 \\ -115000 & 0 & 115000 & 0 & 0 & 0 \\ 0 & 0 & 0 & 76666.67 & 0 & -76666.67 \\ -19628 & -29442 & 0 & 0 & 19628 & 29442 \\ -29442 & -44163 & 0 & -76666.67 & 29442 & 120829.67 \end{bmatrix} \times \begin{Bmatrix} U_1 \\ V_1 \\ U_2 \\ V_2 \\ U_3 \\ V_3 \end{Bmatrix} = \begin{Bmatrix} R_{X1} \\ R_{Y1} \\ 0 \\ R_{Y2} \\ 12000 \\ 0 \end{Bmatrix}$$

or in compact form as

$$[K]\{\delta\} = \{F\}$$

The system of equation can not be solved in a unique fashion since the stiffness matrix is singular.

To solve the system of equations, it is neccessary to partition the matrix $[\mathbf{K}]$ according to known and unknown quantities. The vector of displacements $\{\delta\}$ can be partitioned into known and unknown quantities.

$$\{\delta\} = \begin{Bmatrix} U_1 = 0 \\ V_1 = 0 \\ V_2 = 0 \\ \cdots \\ U_2 \\ U_3 \\ V_3 \end{Bmatrix}$$

Similarly, the right-hand-side vector of global forces can be partitioned accoordingly:

$$\{F\} = \begin{Bmatrix} R_{X1} \\ R_{Y1} \\ R_{Y2} \\ \cdots \\ 0 \\ 12000 \\ 0 \end{Bmatrix}$$

Note that unknown displacements corresponds to know forces correspond and known displacements correspond to unknown forces. Finally, the matrix $[\mathbf{K}]$ is partitioned as:

$$\begin{bmatrix} 134628 & 29442 & 0 & \vdots & -115000 & -19628 & -29442 \\ 29442 & 44163 & 0 & \vdots & 0 & -29442 & -44163 \\ 0 & 0 & 76666.67 & \vdots & 0 & 0 & -76666.67 \\ \cdots & \cdots & \cdots & \cdots & \cdots & \cdots & \cdots \\ -115000 & 0 & 0 & \vdots & 115000 & 0 & 0 \\ -19628 & -29442 & 0 & \vdots & 0 & 19628 & 29442 \\ -29442 & -44163 & -76666.67 & \vdots & 0 & 29442 & 120829.67 \end{bmatrix} \times \begin{Bmatrix} U_1 = 0 \\ V_1 = 0 \\ V_2 = 0 \\ \cdots \\ U_2 \\ U_3 \\ V_3 \end{Bmatrix} = \begin{Bmatrix} R_{X1} \\ R_{Y1} \\ R_{Y2} \\ \cdots \\ 0 \\ 12000 \\ 0 \end{Bmatrix}$$

As a result of the position of $V_2$ being interchanged with that of $U_2$ in the vector $\{\delta\}$, column 3 and line 3 have also been respectively interchanged with column 4 and line 4 in the matrix $[K]$. Finally, the partitioned system of equations can be rewritten in a compact form as

$$\begin{bmatrix} [K_{PP}] & \vdots & [K_{PF}] \\ \cdots & \cdots & \cdots \\ [K_{FP}] & \vdots & [K_{FF}] \end{bmatrix} \begin{Bmatrix} \{\delta_P\} \\ \cdots \\ \{\delta_F\} \end{Bmatrix} = \begin{Bmatrix} \{F_P\} \\ \cdots \\ \{F_F\} \end{Bmatrix}$$

where

The subscripts $P$ and $F$ refer respectively to the prescribed and free degrees of freedom

$\{\delta_P\}^T = \{0.\ 0.\ 0.\}$ the vector of the known prescribed displacements

$\{\delta_F\}^T = \{U_2\ U_3\ V_3\}$ the vector of the unknown free displacements

$\{F_P\}^T = \{R_{X1}\ R_{Y1}\ R_{Y2}\}$ the vector of the unknown reaction forces corresponding to the prescribed displacements

$\{F_F\}^T = \{0\ 12000\ 0\}$ the vector of the known applied external forces

## Solution of the system of equations

Equation can be expanded to yield

$$[K_{PP}]\{\delta_P\} + [K_{PF}]\{\delta_F\} = \{F_P\}$$

$$[K_{FP}]\{\delta_P\} + [K_{FF}]\{\delta_F\} = \{F_F\}$$

Since $\{\delta_P\}$ and $\{F_F\}$ are known quantities, it is then possible to obtain the vector $\{\delta_F\}$ as

$$\{\delta_F\} = [K_{FF}]^{-1}\{\{F_F\} - [K_{FP}]\{\delta_P\}\}$$

However, since $\{\delta_P\}^T = \{0.\ 0.\ 0.\}$,

$$\{\delta_F\} = [K_{FF}]^{-1}\{F_F\}$$

which is simply equivalent to eliminating the lines and the columns corresponding to the restrained degrees of freedom in the global matrix; that is,

$$\begin{Bmatrix} U_2 \\ U_3 \\ V_3 \end{Bmatrix} = \begin{bmatrix} 115000 & 0 & 0 \\ 0 & 19628 & 29442 \\ 0 & 29442 & 120829.67 \end{bmatrix}^{-1} \begin{Bmatrix} 0 \\ 12000 \\ 0 \end{Bmatrix}$$

Solving the system of equations yields

$$\{\delta_F\} = \begin{Bmatrix} U_2 \\ U_3 \\ V_3 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0.9635 \\ -0.2348 \end{Bmatrix} \text{mm}$$

In summary, the vector of global displacements can be obtained as

$$\{\delta\} = \begin{Bmatrix} U_1 = 0. \\ V_1 = 0. \\ U_2 = 0. \\ V_2 = 0. \\ U_3 = 0.9635 \\ V_3 = -0.2348 \end{Bmatrix}$$

## Support reaction

Once $\{\delta_F\}$ is known, it is possible to obtain the vector of the unknown reaction forces $\{F_P\}^T = \{R_{X1}\ R_{Y1}\ R_{Y2}\}$. Since $\{\delta_P\}^T = \{0.\ 0.\ 0.\}$, the vector $\{F_P\}$ is obtained as

$$\{F_P\} = [K_{PF}]\,\{\delta_F\}$$

That is,

$$\begin{Bmatrix} R_{X1} \\ R_{Y1} \\ R_{Y2} \end{Bmatrix} = \begin{bmatrix} -115000 & -19628 & -29442 \\ 0 & -29442 & -44163 \\ 0 & 0 & -76666.67 \end{bmatrix} \begin{Bmatrix} 0 \\ 0.9635 \\ -0.2348 \end{Bmatrix} = \begin{Bmatrix} -12 \\ -18 \\ 18 \end{Bmatrix} \text{kN}$$

The obtained values for the support reactions can be easily checked using the equilibrium equations of a rigid body. Considering the free body diagram of the truss as shown in the above, and taking moments with respect to node 1 yields

$$\Sigma_{/1} = R_{Y2} \times 4 - 12 \times 6 = 0 \Longrightarrow R_{Y2} = 18 \text{ kN}$$

Considering vertical equilibrium yields

$$\Sigma_Y = R_{Y2} + R_{Y1} = 0 \Longrightarrow R_{Y1} = -18 \text{ kN}$$

Considering horizontal equilibrium yields

$$\Sigma_X = 12 + R_{X1} = 0 \Longrightarrow R_{X1} = -12 \text{ kN}$$

## Member's Force

Once all the displacements are known, the member forces can be easily obtained. For example, element 3 has the following vector of global displacements, $\{\overline{d_3}\}$, extracted from the global displacements vector $\{\delta\}$ :

$$\{\overline{d_3}\} = \begin{Bmatrix} U_1 = 0 \\ V_1 = 0 \\ U_3 = 0.9635 \\ V_3 = -0.2348 \end{Bmatrix}$$

The vector of displacements in local coordinates $\{d_3\}$ is obtained using the inverse transformation $\{d_3\} = [C_3]^T\{\overline{d_3}\}$; that is,

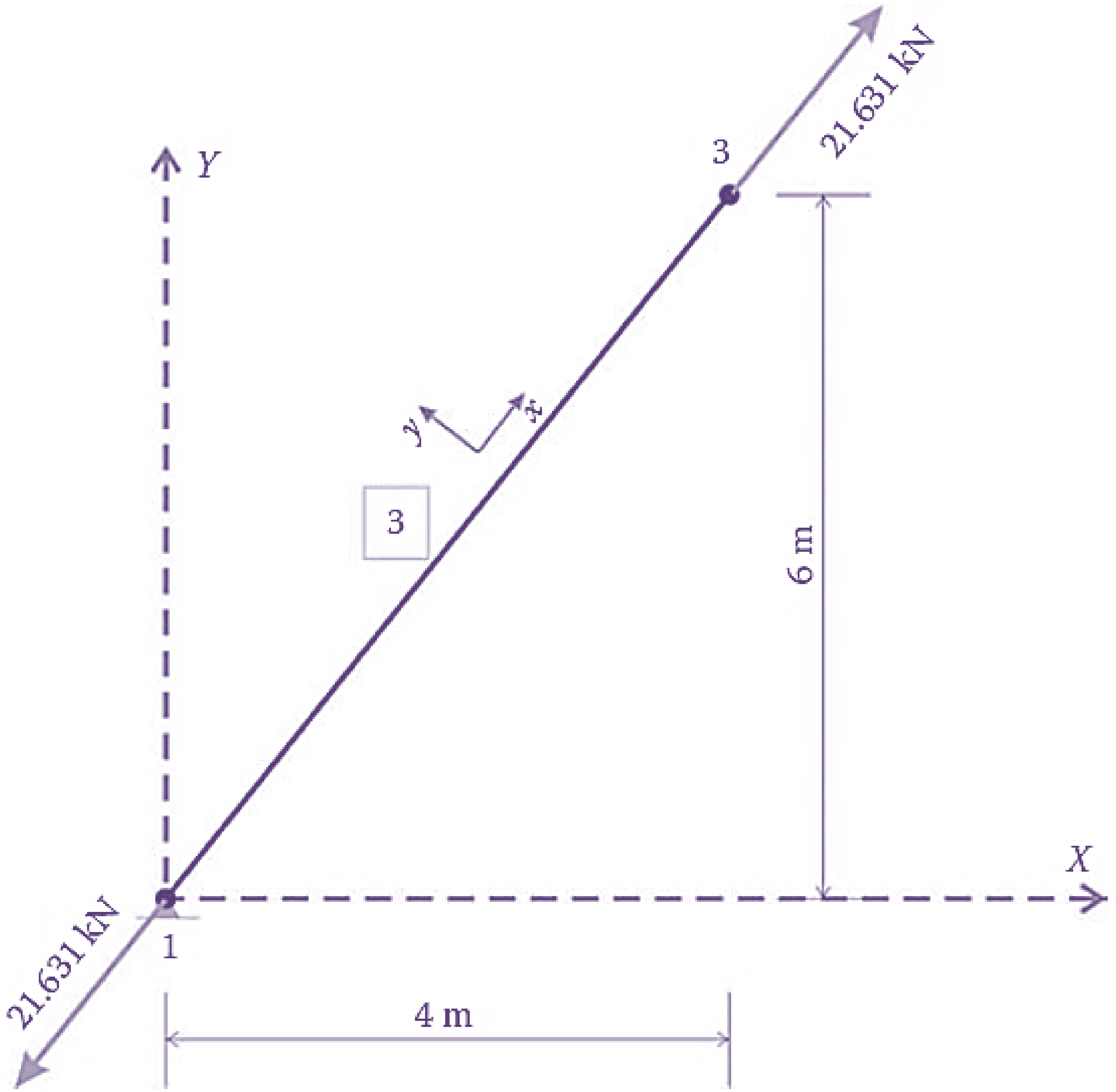
$$\{d_3\} = \begin{bmatrix} 0.554699 & 0.832051 & 0 & 0 \\ -0.832051 & 0.554699 & 0 & 0 \\ 0 & 0 & 0.554699 & 0.832051 \\ 0 & 0 & -0.832051 & 0.554699 \end{bmatrix} \begin{Bmatrix} 0 \\ 0 \\ 0.9635 \\ -0.2348 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \\ 0.3391 \\ -0.9319 \end{Bmatrix}$$

Multiplying the local stiffness matrix of element 3, $[K_3]_L$, by the local displacement vector $\{d_3\}$ yields the local vector of forces $\{f_3\}$; that is,

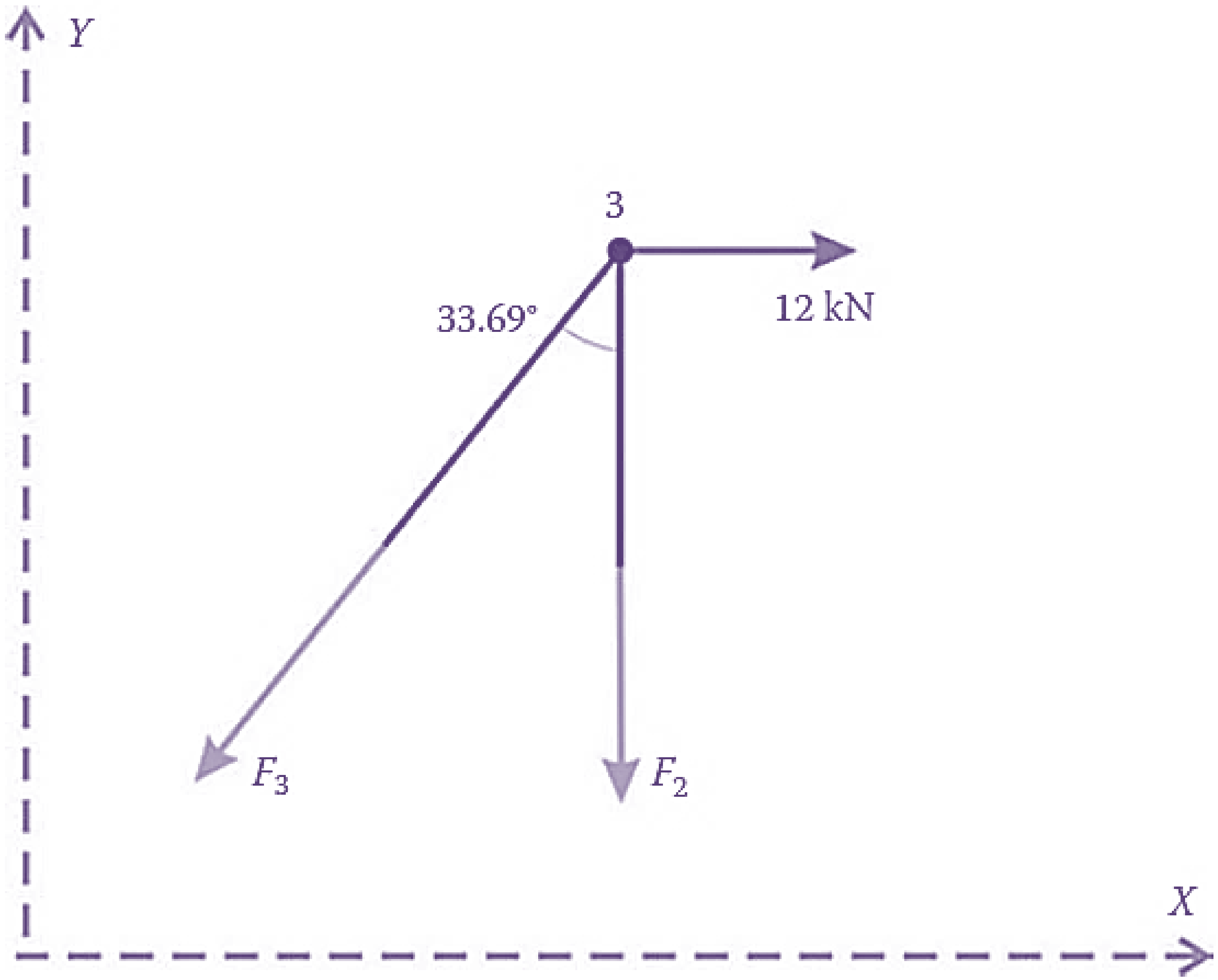
$$\{f_3\} = \begin{bmatrix} 63791.43 & 0 & -63791.43 & 0 \\ 0 & 0 & 0 & 0 \\ -63791.43 & 0 & 63791.43 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \begin{Bmatrix} 0 \\ 0 \\ 0.3391 \\ -0.9319 \end{Bmatrix} = \begin{Bmatrix} -21.631 \\ 0 \\ 21.631 \\ 0 \end{Bmatrix} \text{kN}$$

## Member's Force

The forces on the bar element are represented graphically in the above . It can be seen that the member is under a tensile force of 21.631 kN. This result can be checked using the method of joints.




Free body diagram of element 3



Equilibrium of node 3.

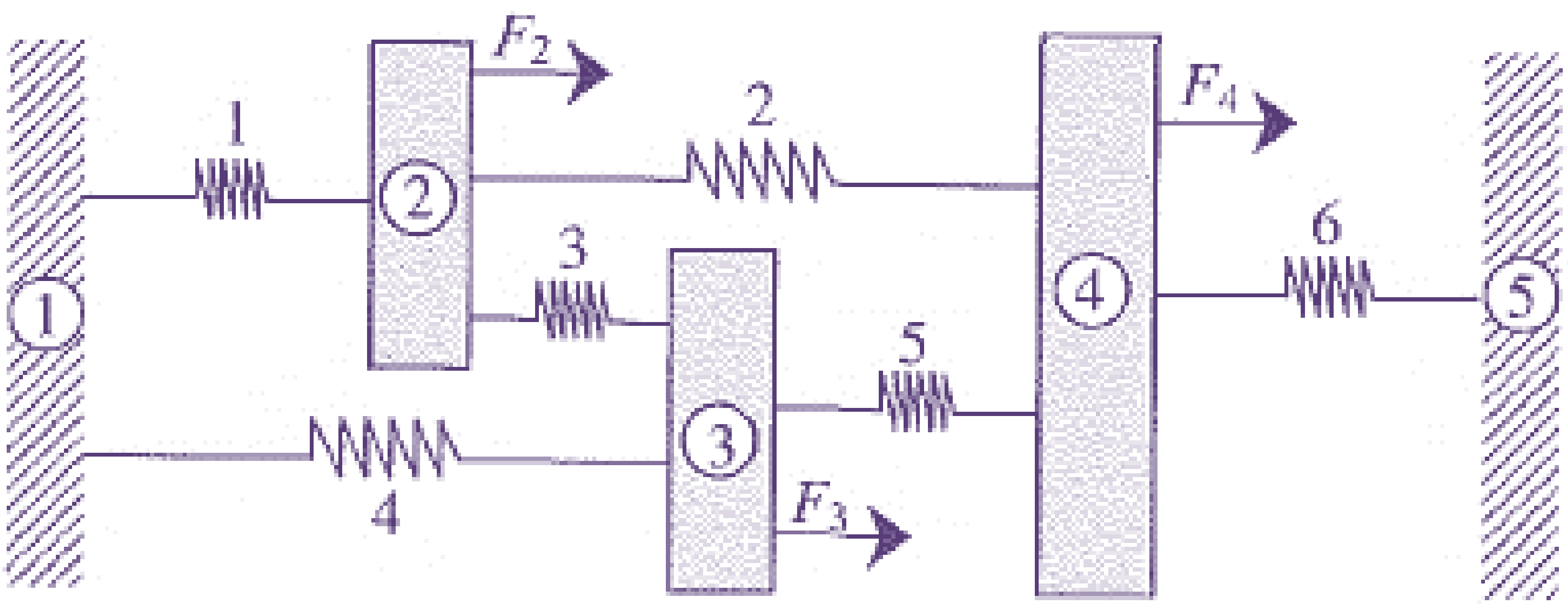
Consider the free body diagram of node (joint) 3. The equilibrium of the joint in the $x$ direction requires

$$\Sigma_X = 12 - F_3 \times \sin(33.69) = 0 \Longrightarrow F_3 = 21.633 \text{ kN}$$

This confirms the obtained result with the finite element method.

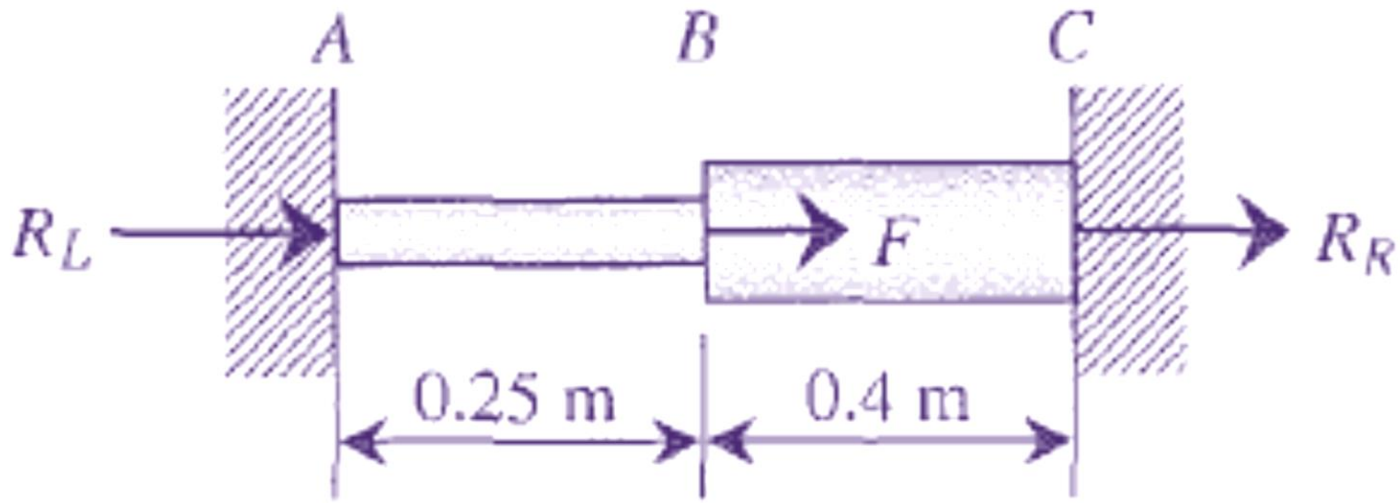
## Homework

1- Consider a system of rigid body connected by springs as shown in Figure. The bodies move only in the horizontal direction. The mass effecs will be ignored. The objective is to determine the stiffness matrix.

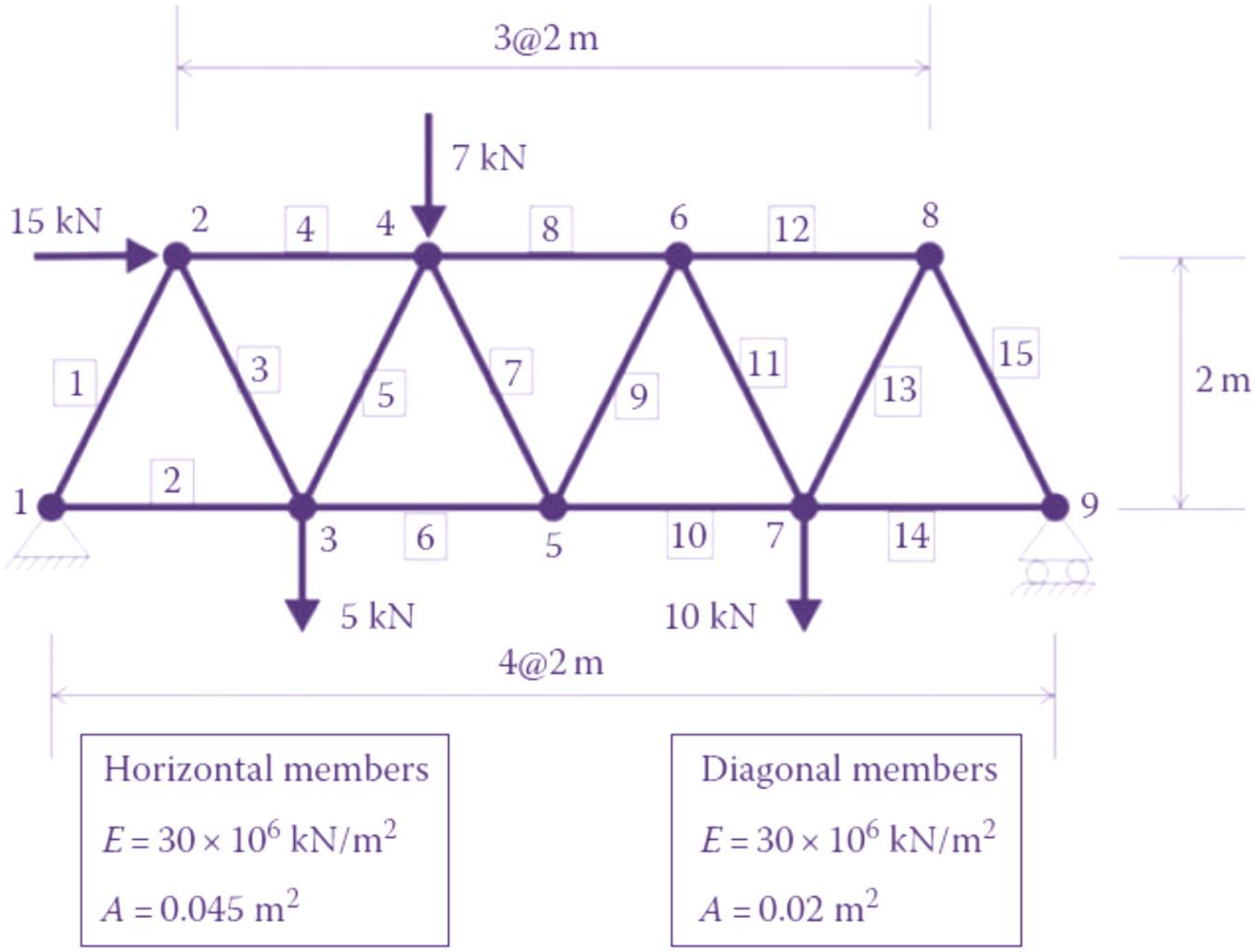


**Homework**

2- Use FEM to determine the axial force **P** in each portion, AB and BC of the uniaxial bar shown in Figure. What are the support reactions? Young's modulus is $E = 100\ GPa$; areas of cross-sections of the two portions AB and BC are $1 \times 10^{-4}\ m^2$ and $2 \times 10^{-4}\ m^2$, repectively. The force F=10000N is applied at the cross-section at B.



# Analysis of a simple truss with Abaqus




Model of Problem

# Modeling

Click **Start**, **All Programs** and locate **Abaqus** as shown in Figure 1

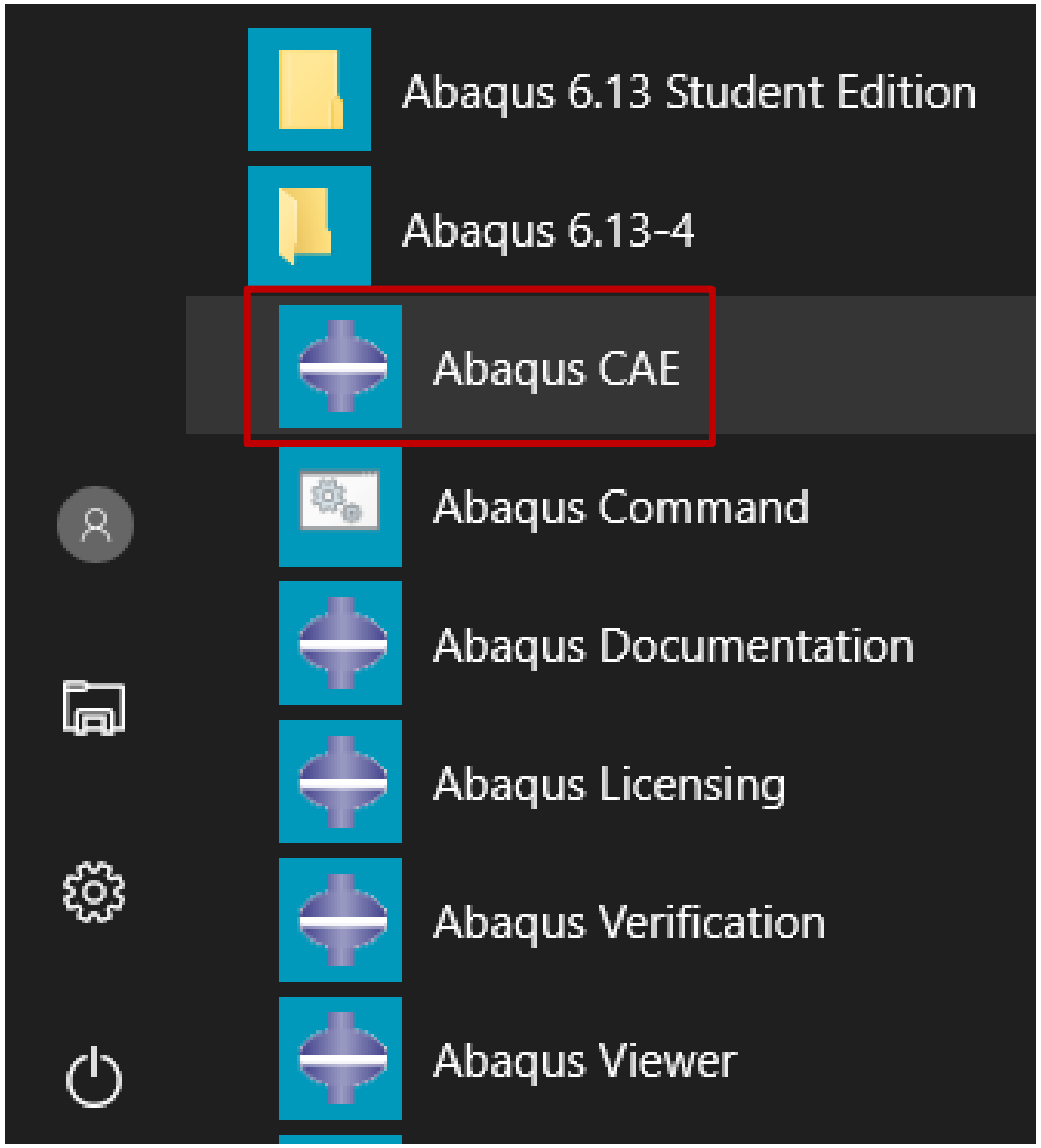



FIGURE 1 Starting Abaqus

Double click on **Abaqus CAE** to reveal the main user interface. Click on **Create Model Database** to start a new analysis. On the main menu, click on **File** and **Set Work Directory** to choose your working directory. Click on **Save As** and name the file **Truss.cae**


**FIGURE 2** Abaqus CAE main user interface

On the left-hand-side menu, click on **Part** to begin creating the model

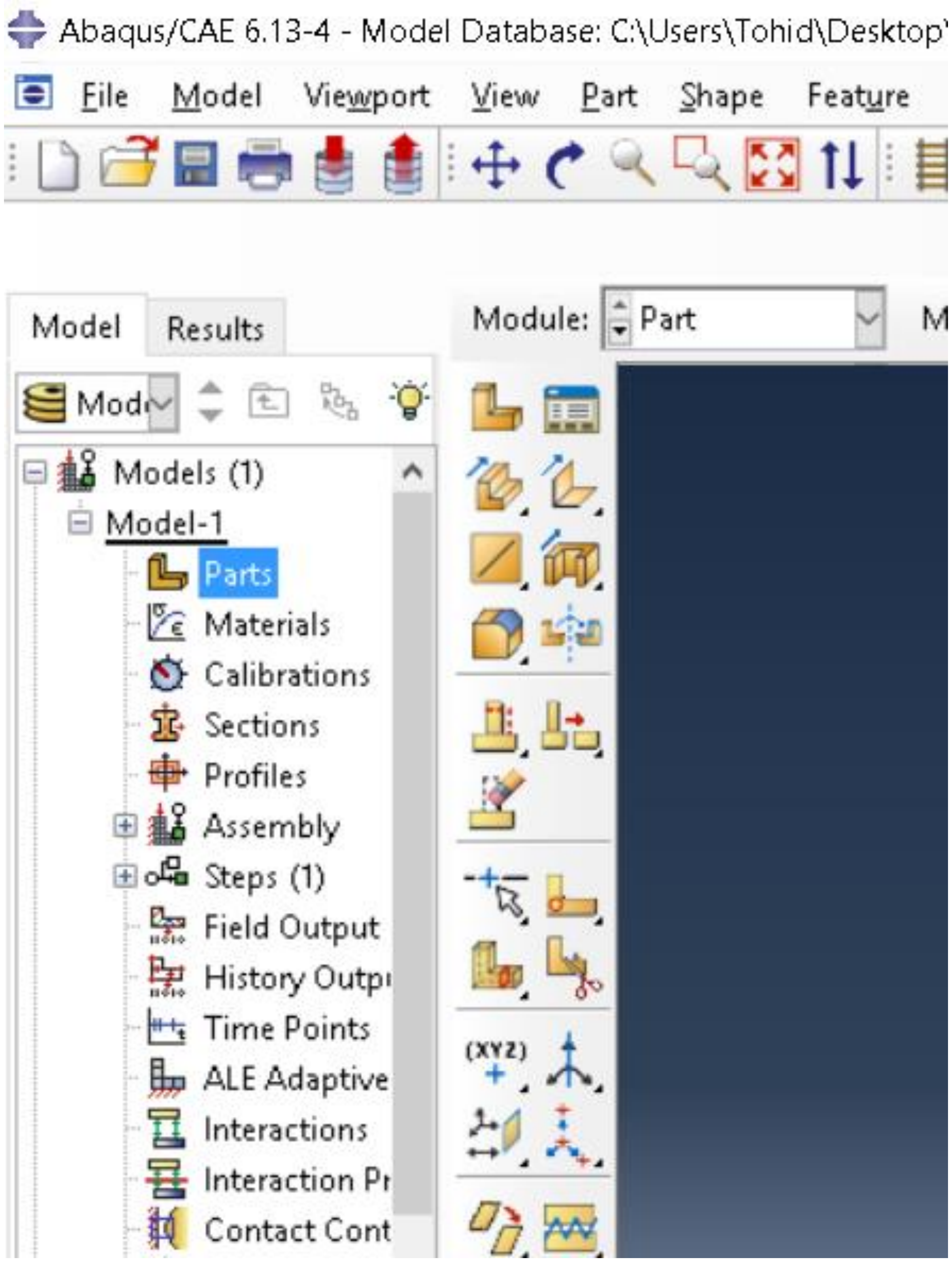



FIGURE 3 Creating a part.

The creating part window shown in Figure 3 appears on the screen. Name the part **Truss_part**, and check **2D Planar** as this is a planar truss, check on **Deformable** in the type. Choose **Wire** as the base feature. Enter an approximate size of 10 m and click on **Continue**. *WARNING: There are no predefined system of units within Abaqus, so the user is responsible for ensuring that the correct units are specified.*

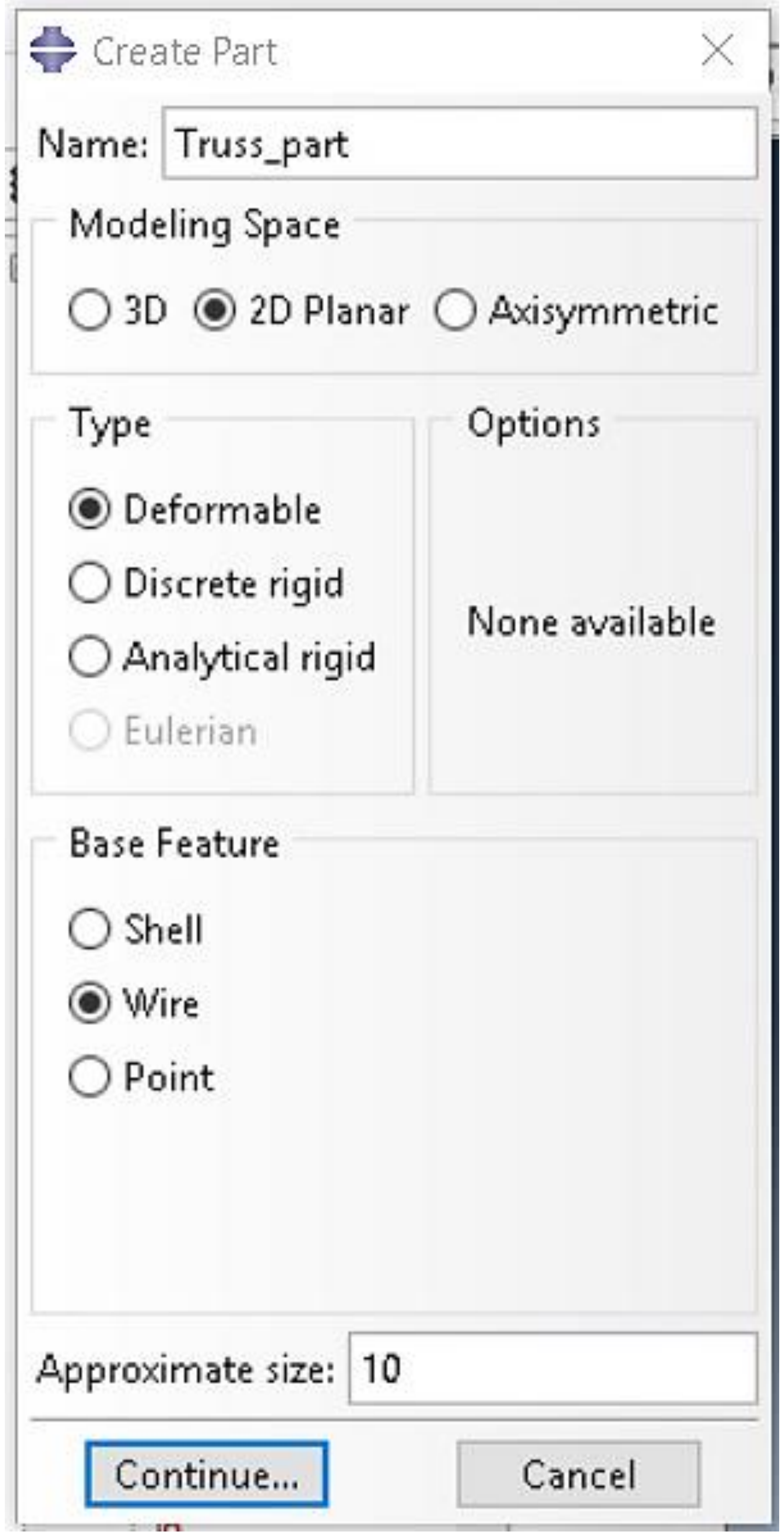



**FIGURE 3** Choosing the geometry of the part.

Click on **Auto-fit View** to fit the view of the sketcher to the screen. You can also place the cursor on the center of the sketcher and zoom in and out using the middle mouse

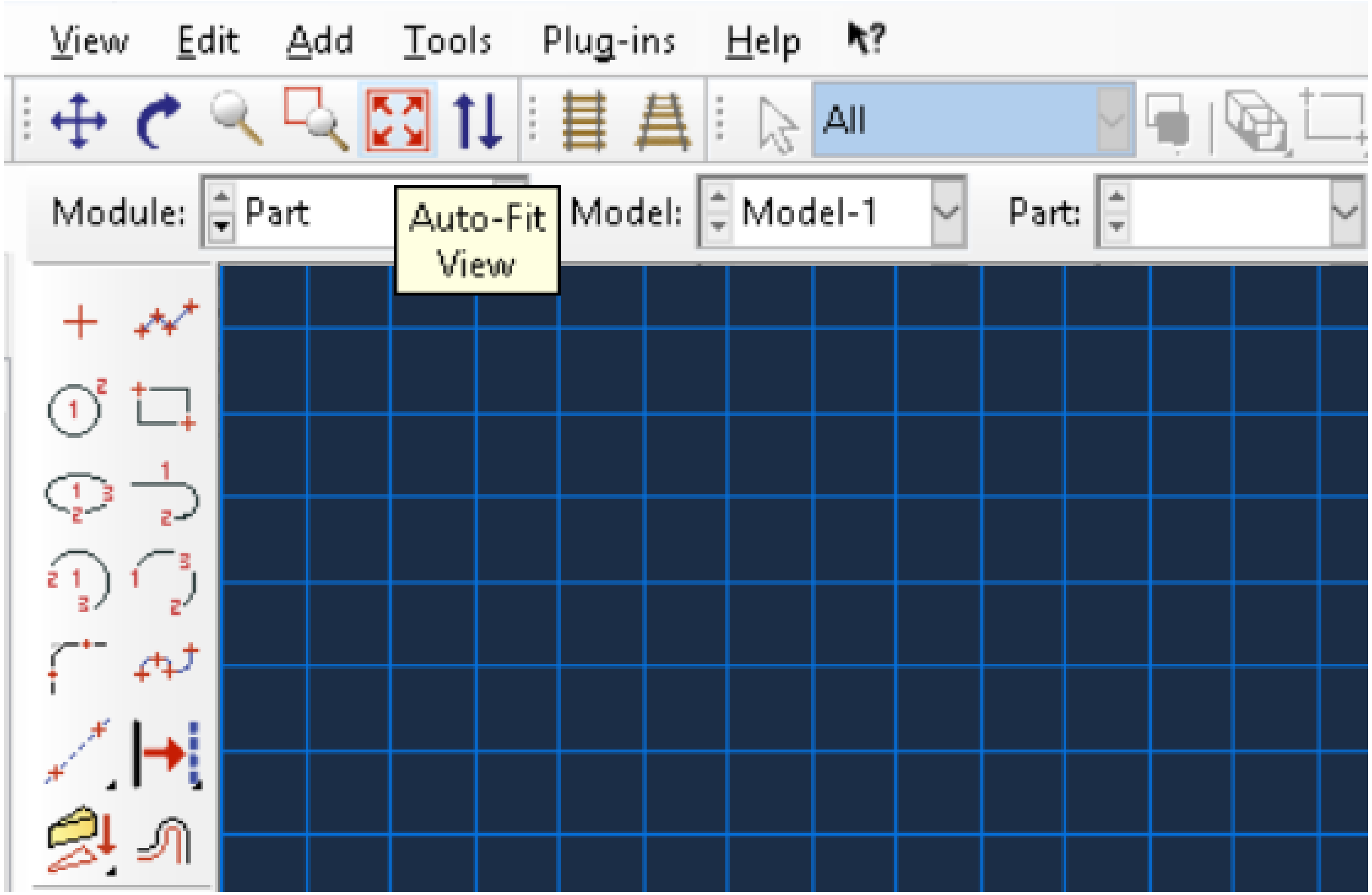



**FIGURE 4** Fitting the sketcher to the screen.

In the sketcher menu, choose the **Create-Lines Connected** button to begin drawing the geometry of the truss

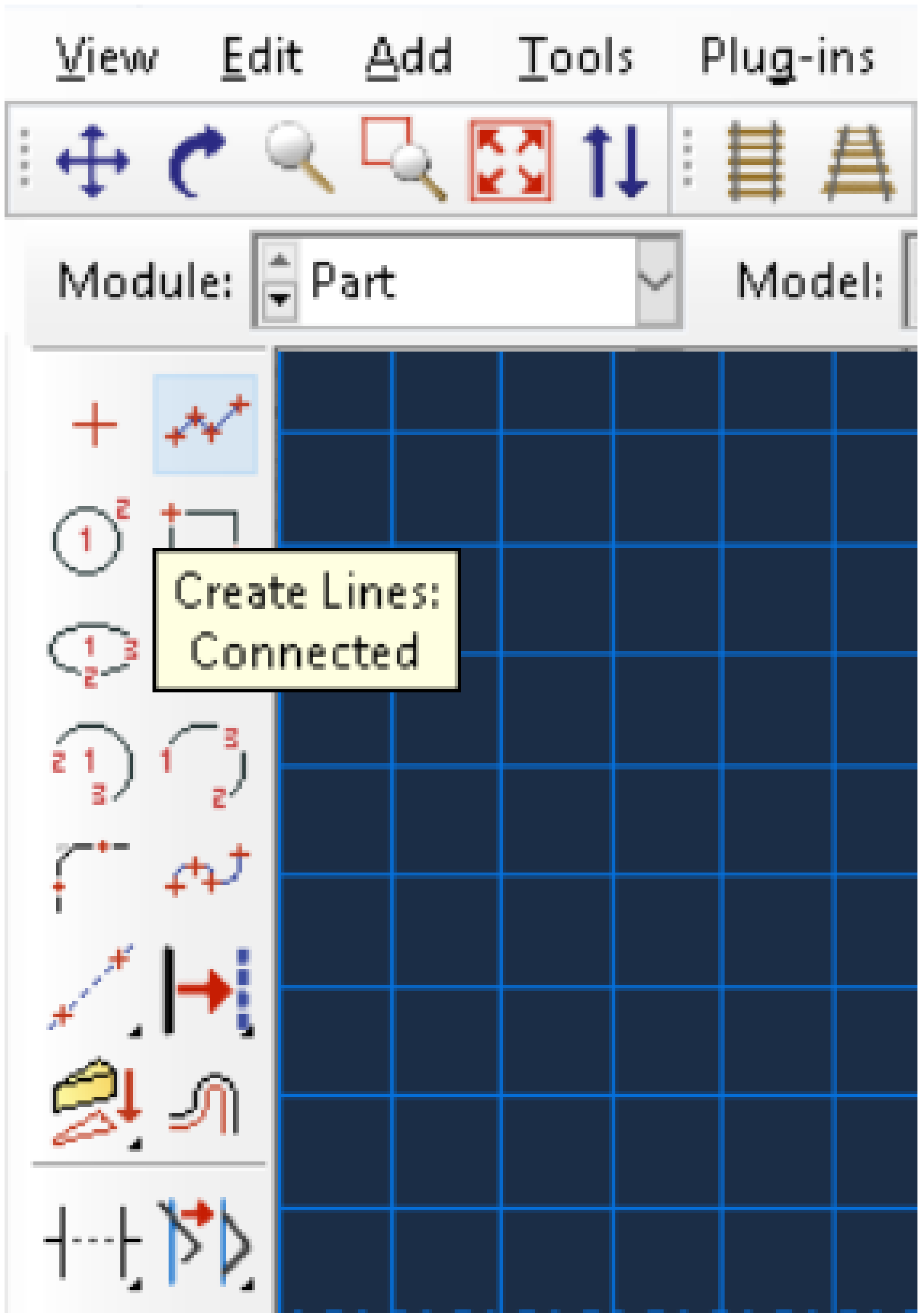



**FIGURE 5** Drawing using the connected line button.

Begin drawing the truss. The coordinates of the cursor are given in the top-left corner. You could also enter them using the **Pick a point or enter X-Y coordinates** in the box situated in the bottom-left corner. Once finished, click on **Done** in the bottom-left corner to exit the sketcher

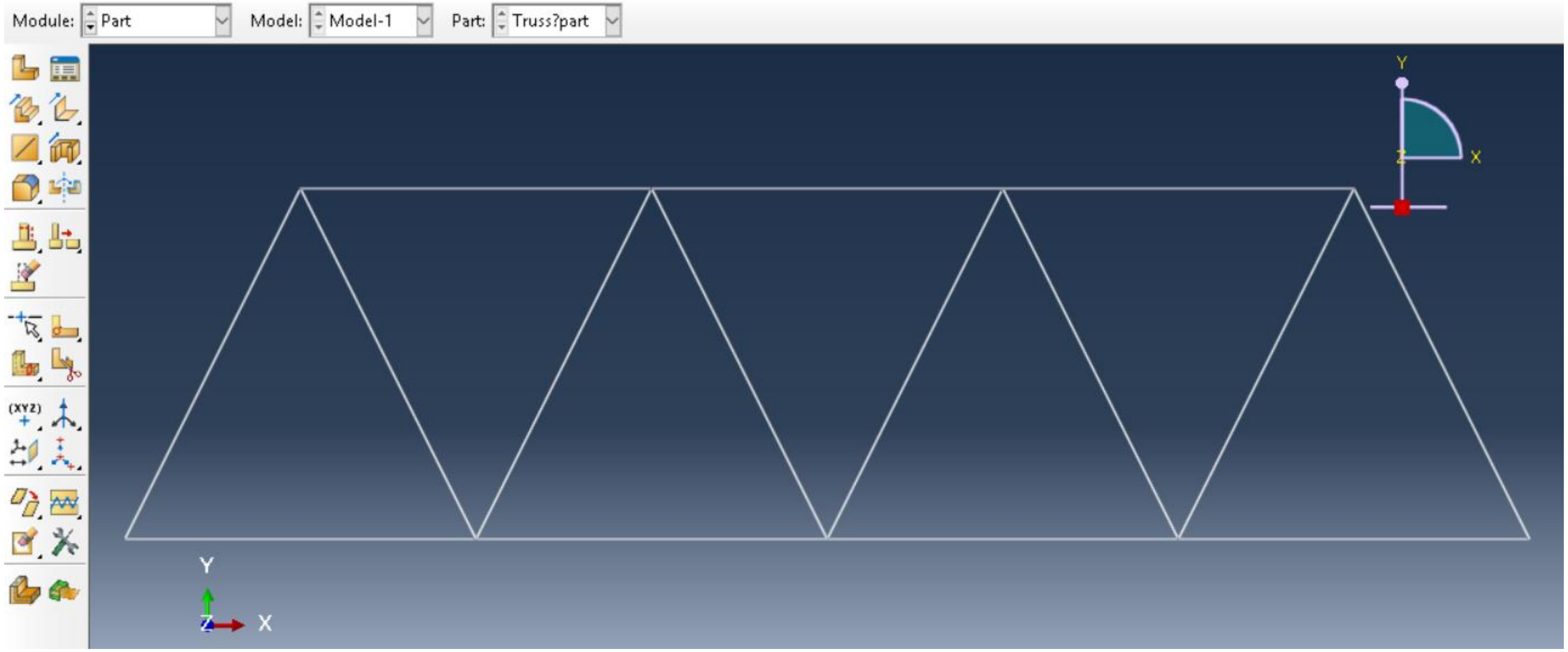
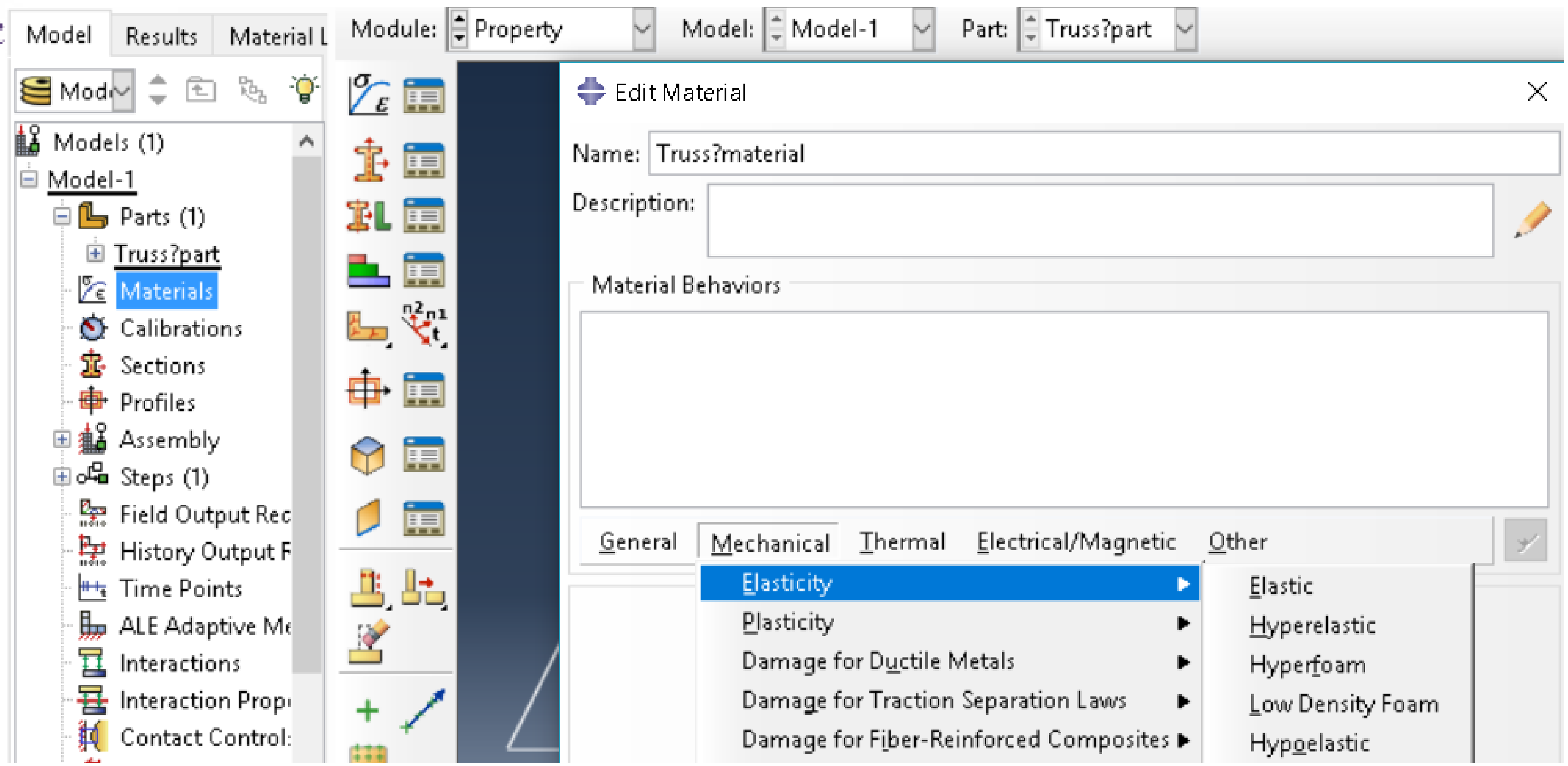



**FIGURE 6** Drawing the truss geometry.

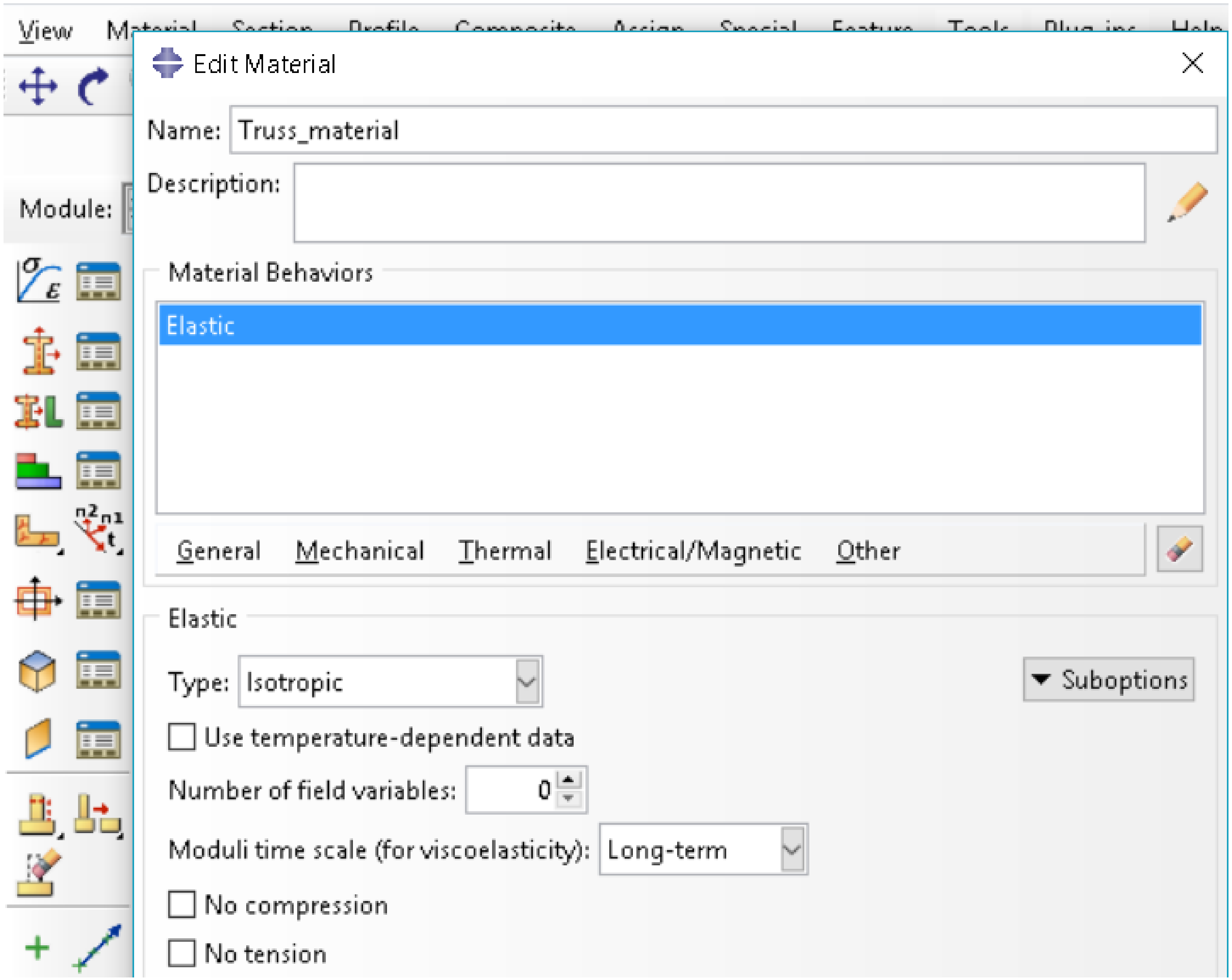
Next, under the model tree, click on **Materials** to create a material for the truss. Since all the members of the truss are made of the same material, we will only define one material, which we will name **Truss_material**. Then click on **Mechanical**, then **Elasticity**, and **Elastic**

FIGURE 7 Material definition



Enter $30.e6\ \mathrm{kN/m^2}$ for the elastic modulus, and 0.3 for Poisson's ratio even though it is not applicable for a truss

**FIGURE8** Material properties.



The longitudinal members of the truss have a cross area of 0.045 $m^2$ and the diagonal members have a cross area of 0.02 $m^2$. To input this data, we need to define two sections

Under the Model tree, click on **Sections** and the **Create Section** window appears. Name the section **Longitudinal**. In the **Category** check **Beam**, and in the **Type**, choose **Truss**. Click on **Continue**.

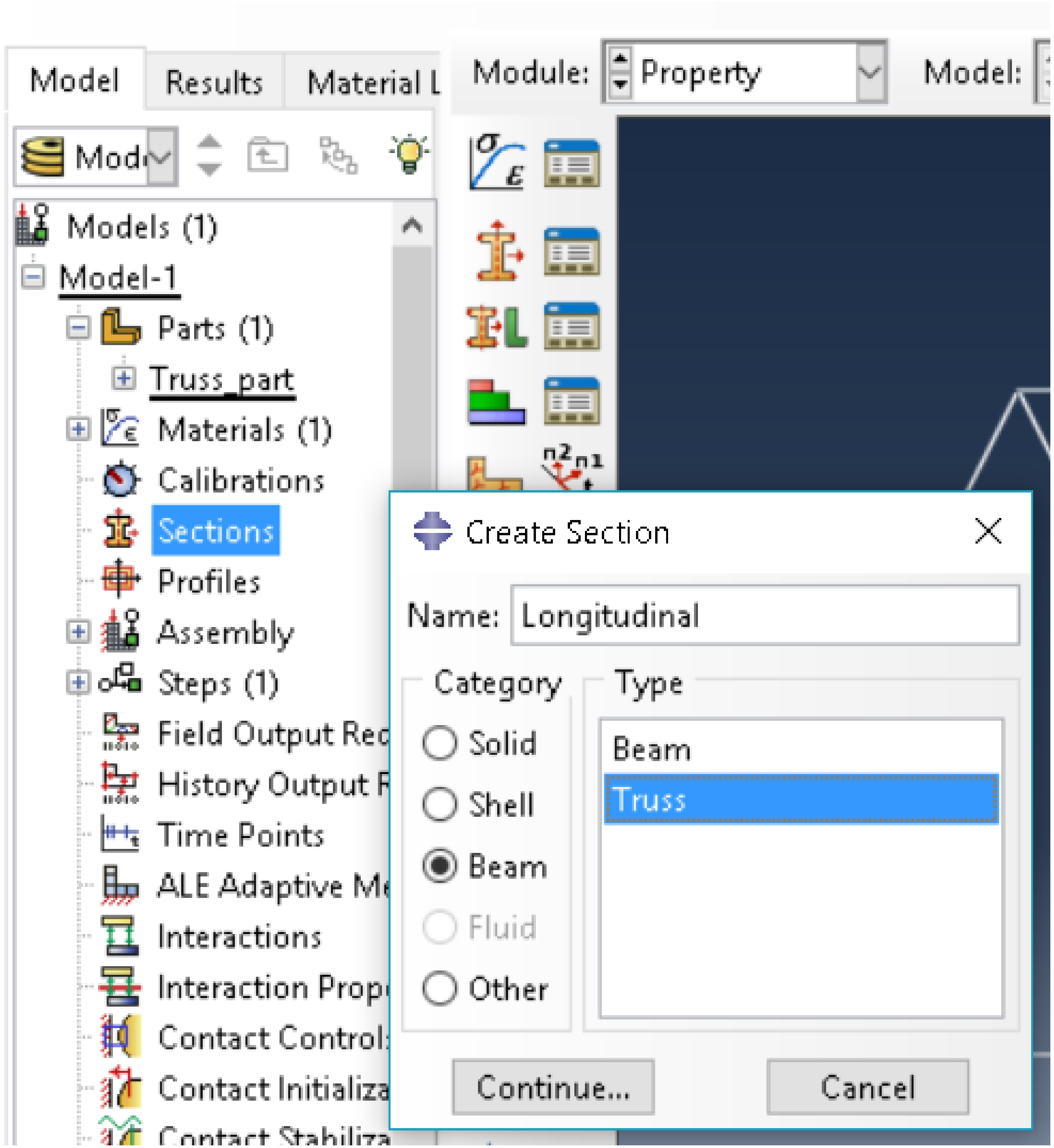



**FIGURE 9** Create section window.

Next the **Edit Section** window appears. Scroll through **Material** and choose the already created material **Truss_material** to assign it to the section. In **Cross sectional area** enter 0.045 $m^2$ and click **OK**

Follow exactly the same procedure to create another section named **Diagonal** and enter 0.02 $m^2$ for the cross area.

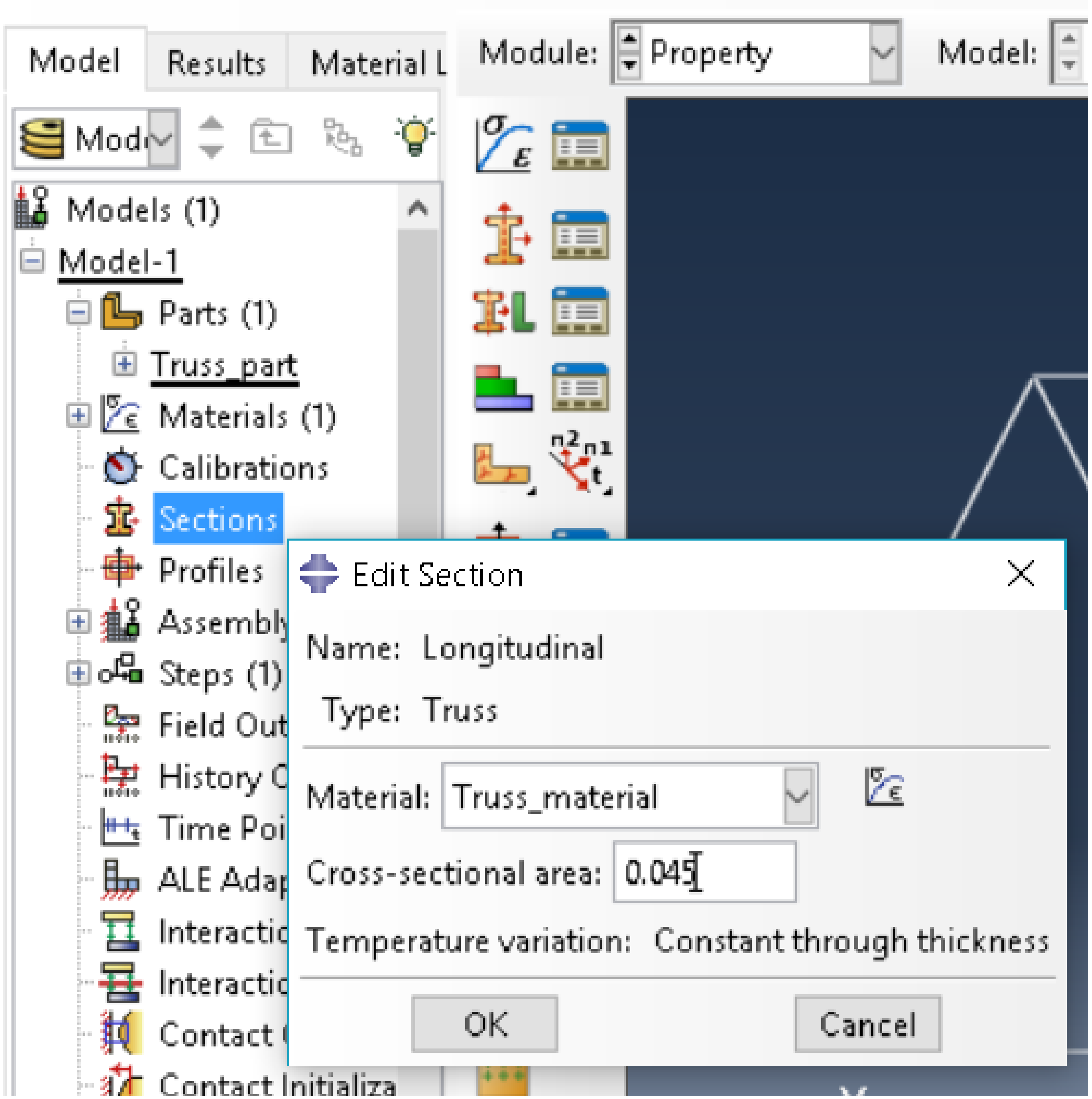



FIGURE 10 Edit material window.

Next we assign the defined sections to the corresponding members. Expand the menu under **Truss_part** and click on **Section assignment**. The message **Select the regions to be assigned a section** should appear on the bottom-left corner of the main window

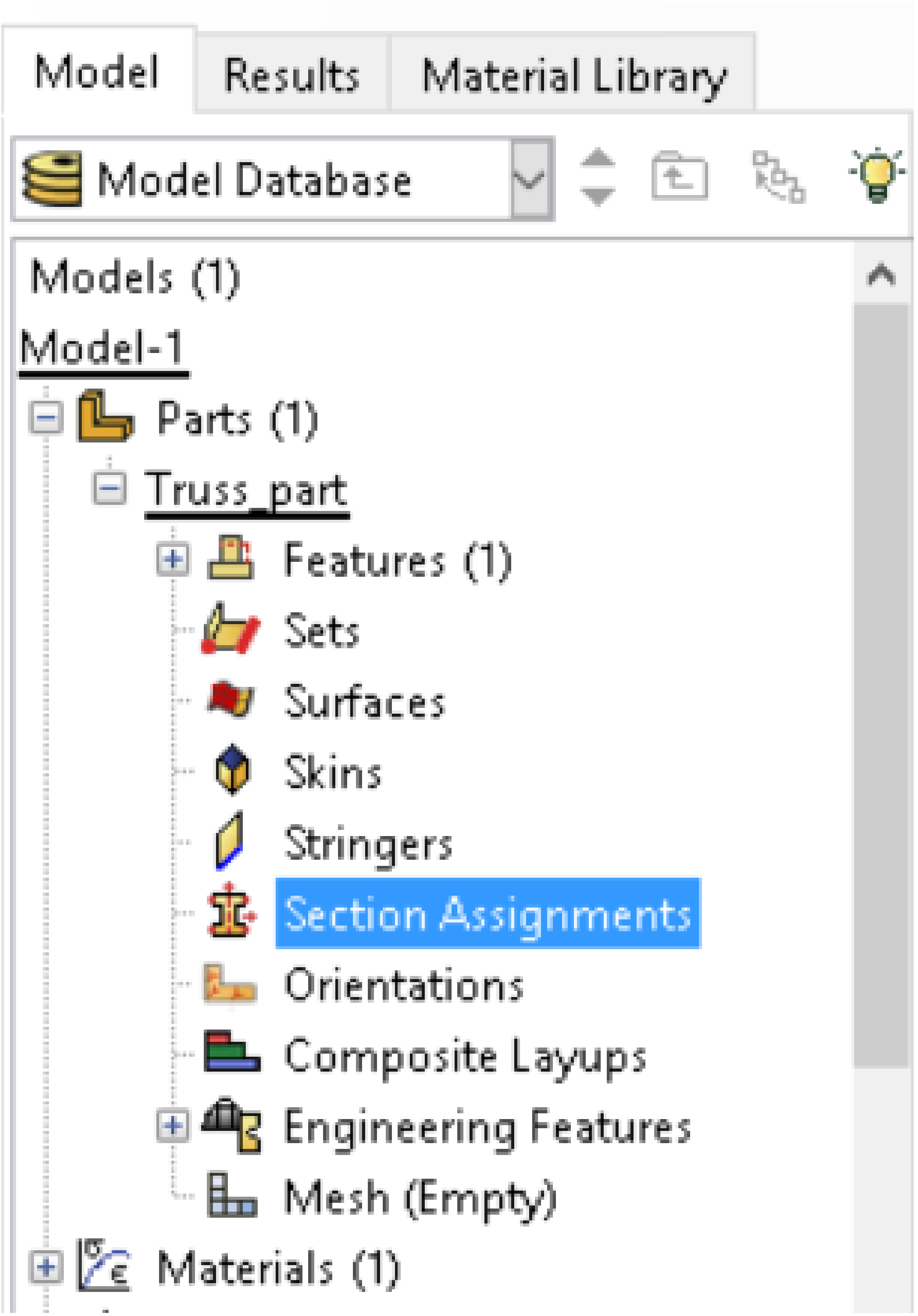



FIGURE 11 Section assignment.

Keep the Shift key down, and with the mouse select the horizontal members. Once a member is selected it changes color. Click on **done** in the bottom-left corner next to the message **Select the regions to be assigned a section**. The **Edit Section Assignment** window appears

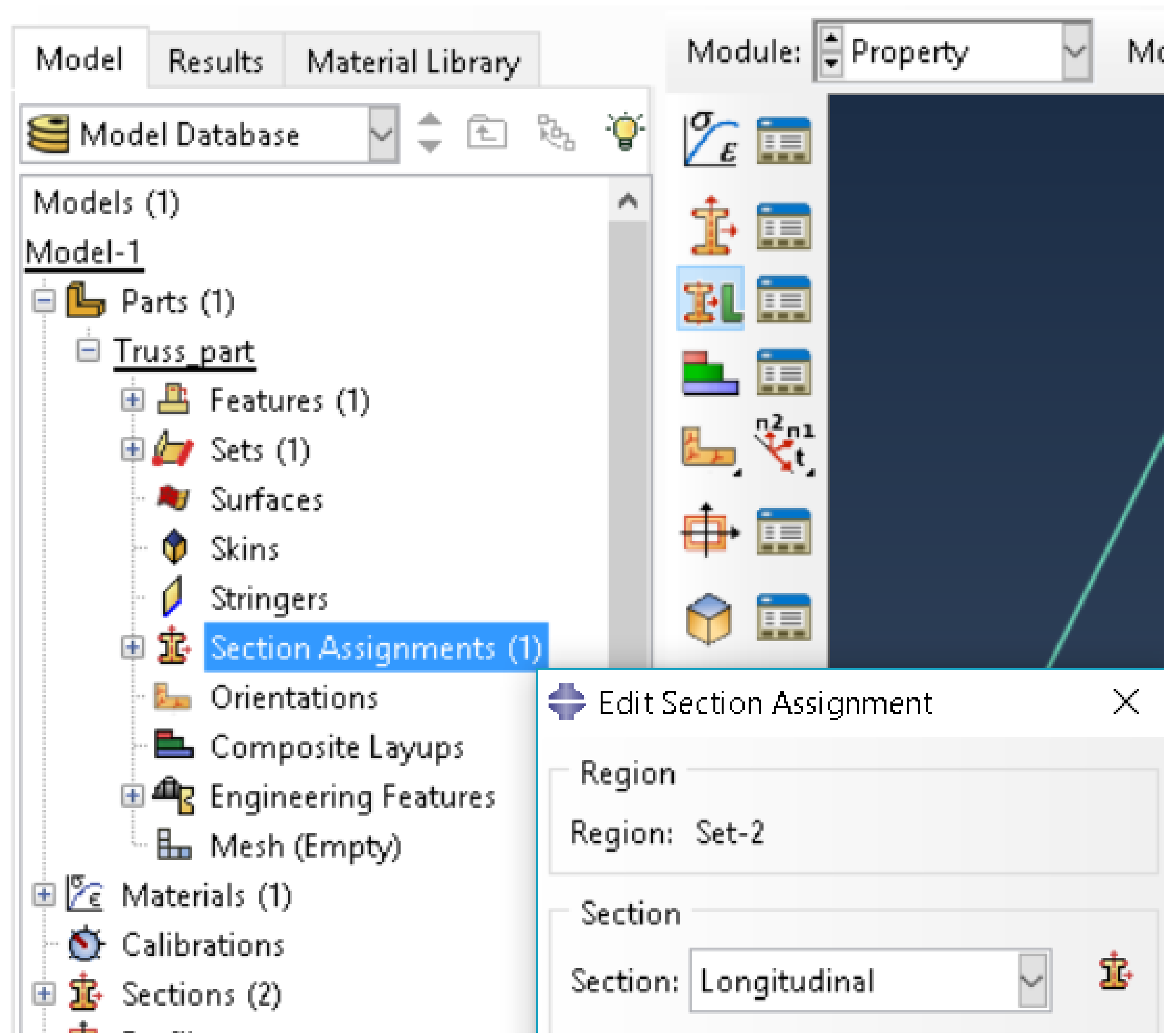



**FIGURE 12** Edit section assignment.

In the next step, we will define the elements. Expand the menu under **Truss_part** and click on **Mesh(empty)** to load the meshing menu

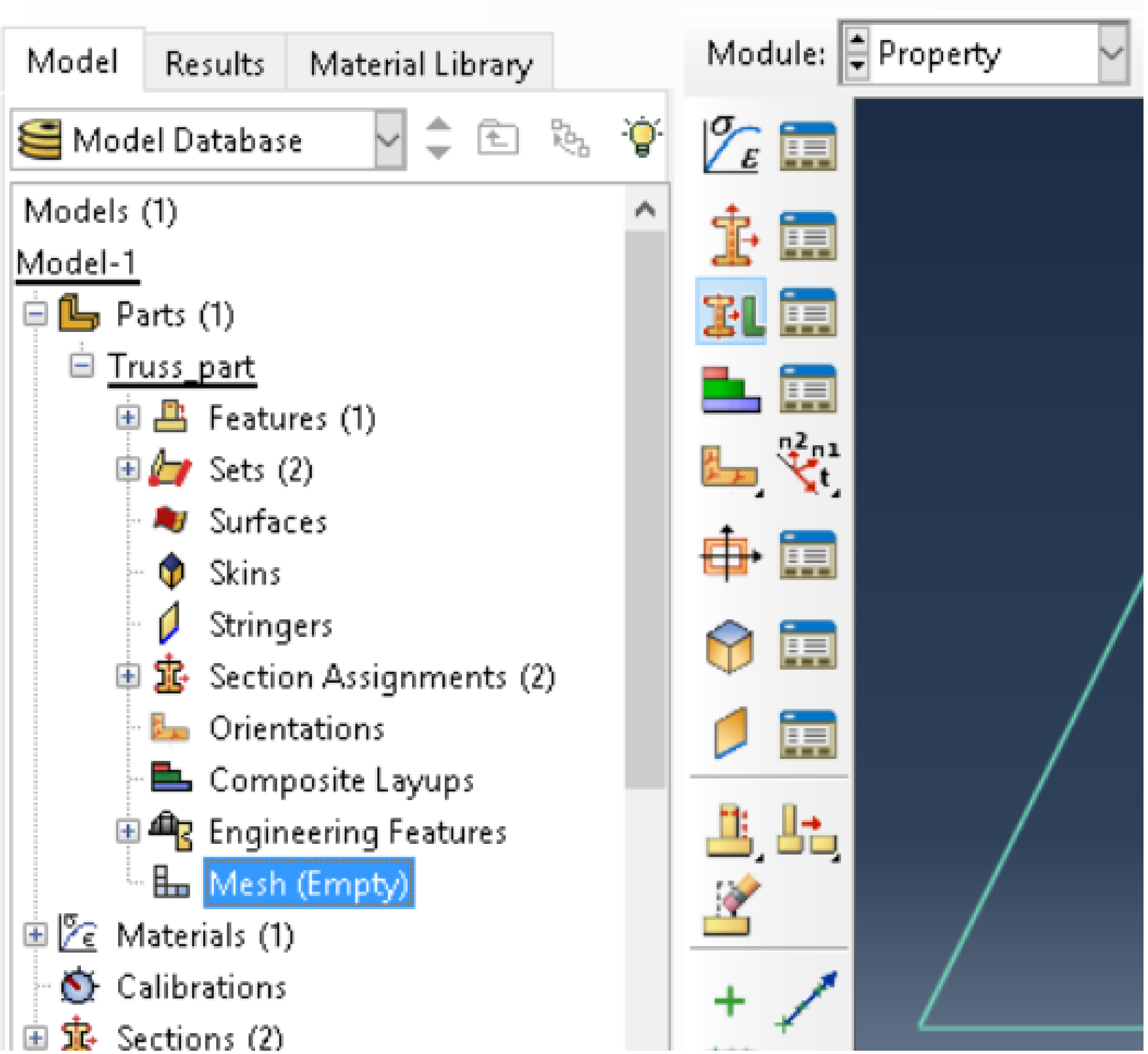



FIGURE13 Loading the meshing menu.

On the main menu, click on **Mesh** and then on **Element Type**, and with the mouse select the whole truss. Click on **Done** in the bottom-left corner of the main window

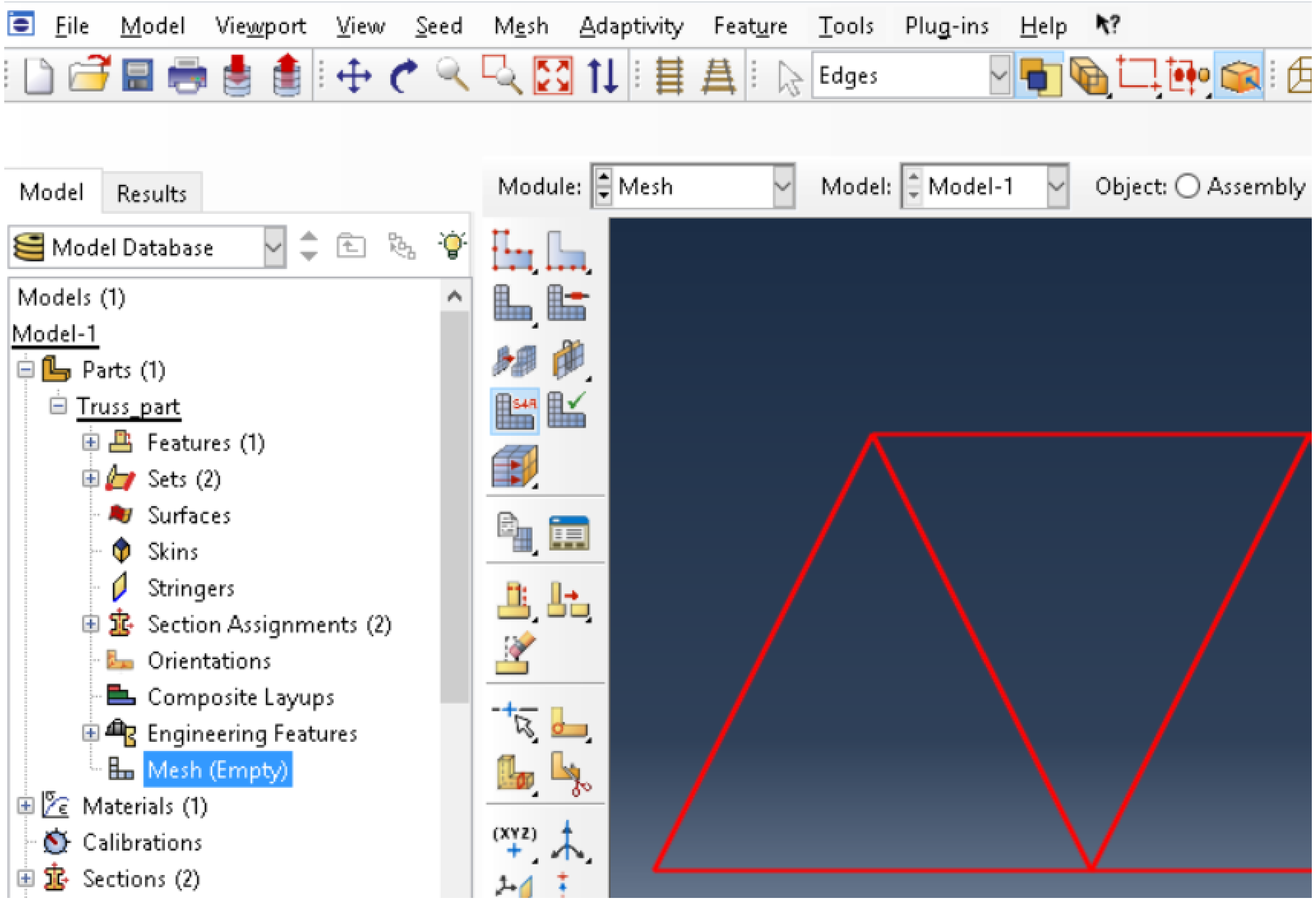
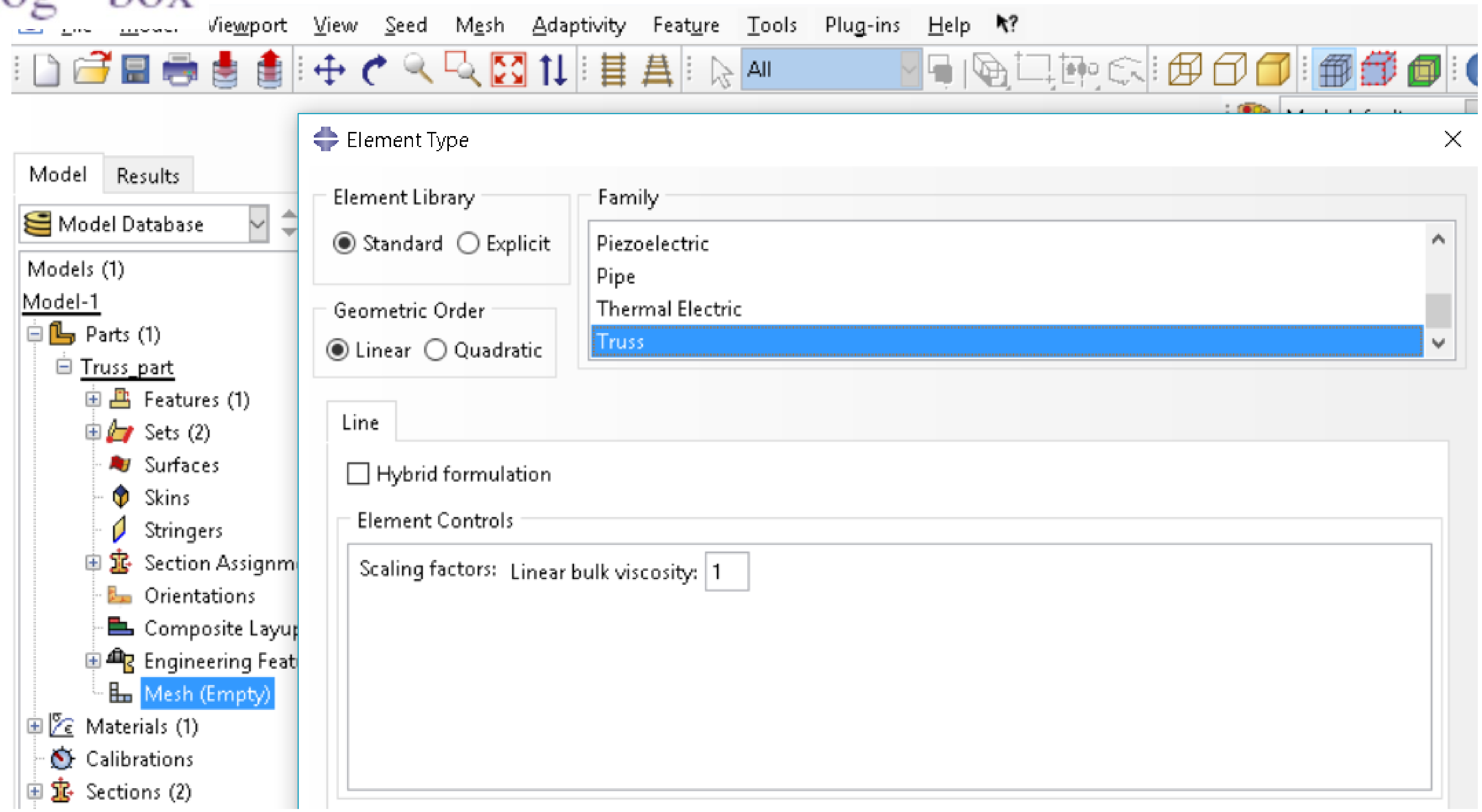



**FIGURE14** Selecting regions to be assigned element type.

The element type dialog box appears. In **Element Library** click on **Standard**. In **Element family** scroll down and choose **Truss**. In **Geometric order**, choose **Linear**. The message **T2D2: A 2: node linear 2-D truss** should appear in the dialog box

FIGURE 15 Selecting element type.



On the main menu click on **Seed**, then on **Edge by number**, and select the whole truss. Enter 1 in the bottom-left corner of the main window, and press **Enter**. The seeding on the truss should look like Figure

On the main menu, click on **Mesh** again, and then on **Part** to mesh the truss. Once meshed, the truss changes color to blue.

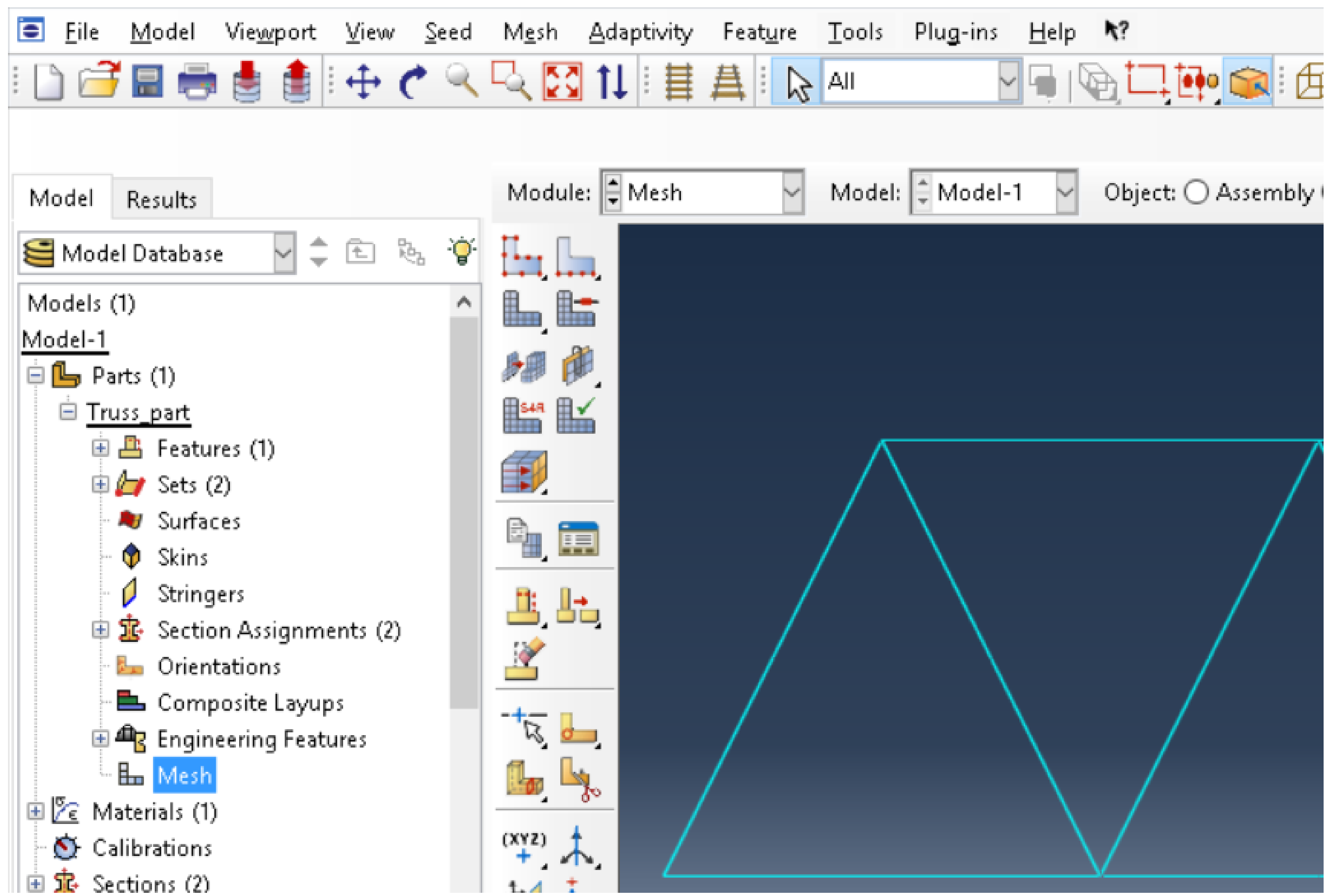



**FIGURE 16** Mesh.

Expand the menu under **Assembly** and double click on **instances**.

**The create instance** dialog box appears. In this case, we have only one part: **Truss_part**. Select it and click **OK**

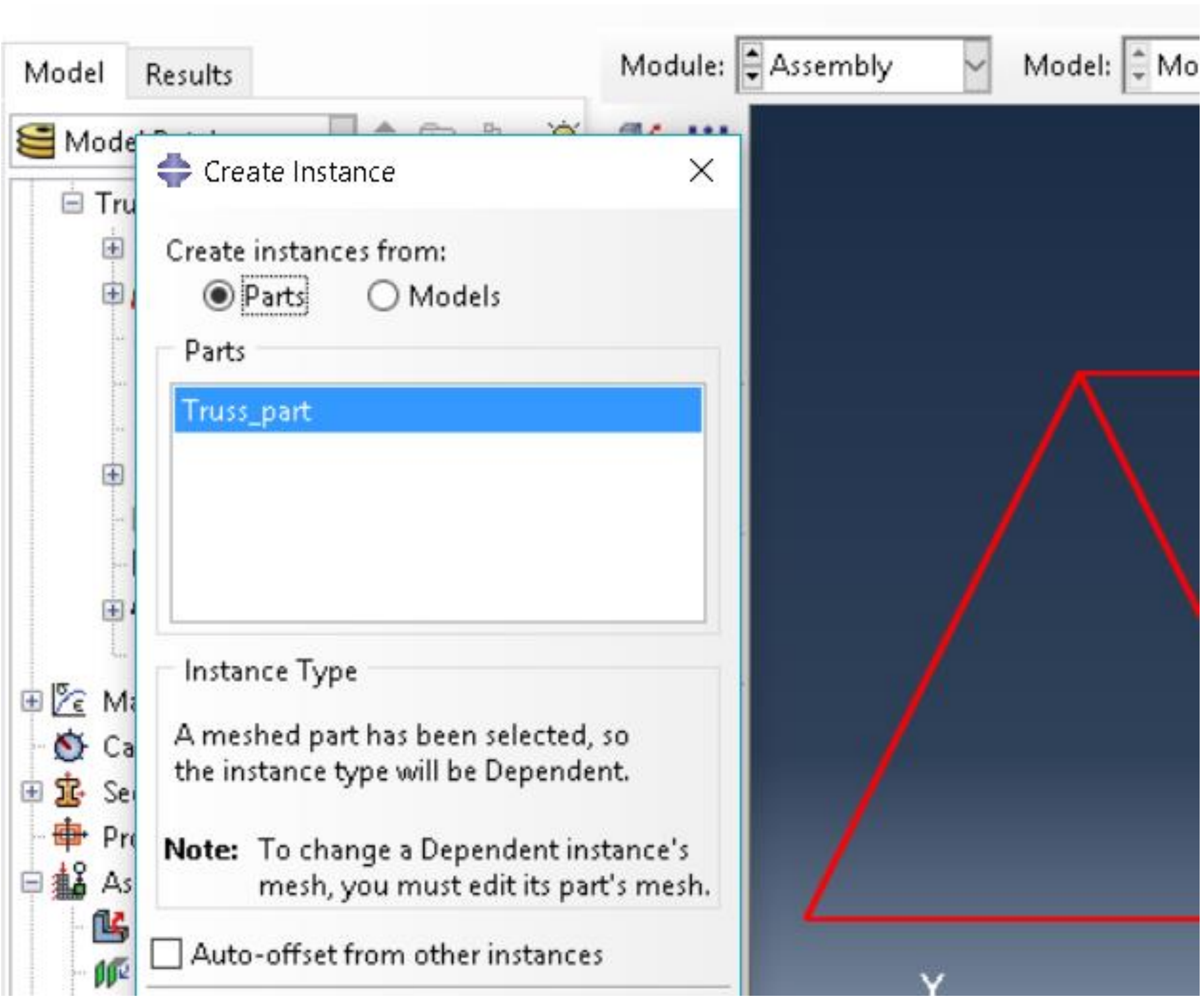
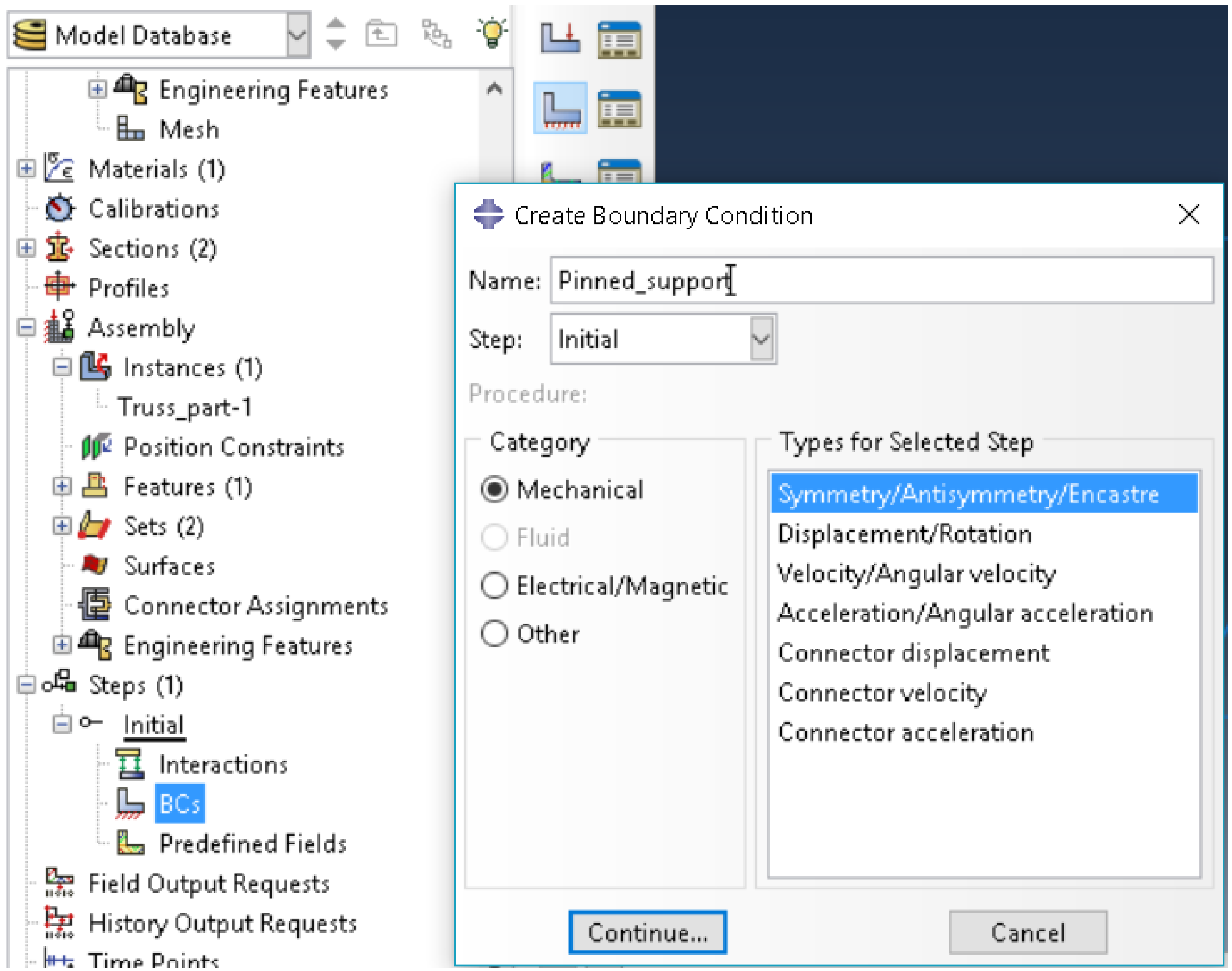



FIGURE 17 Creating instances.

Expand the menu under **Steps** and **Initial**, click on **BC** to introduce the boundary conditions

The **Create Boundary Condition** dialog box appears. Name the boundary condition **Pinned_support**. Choose **Symmetry/Antisymmetry/Encastré** and click on **Continue**

FIGURE 18 Type of boundary conditions.



Select the left-side support and click on **Done**

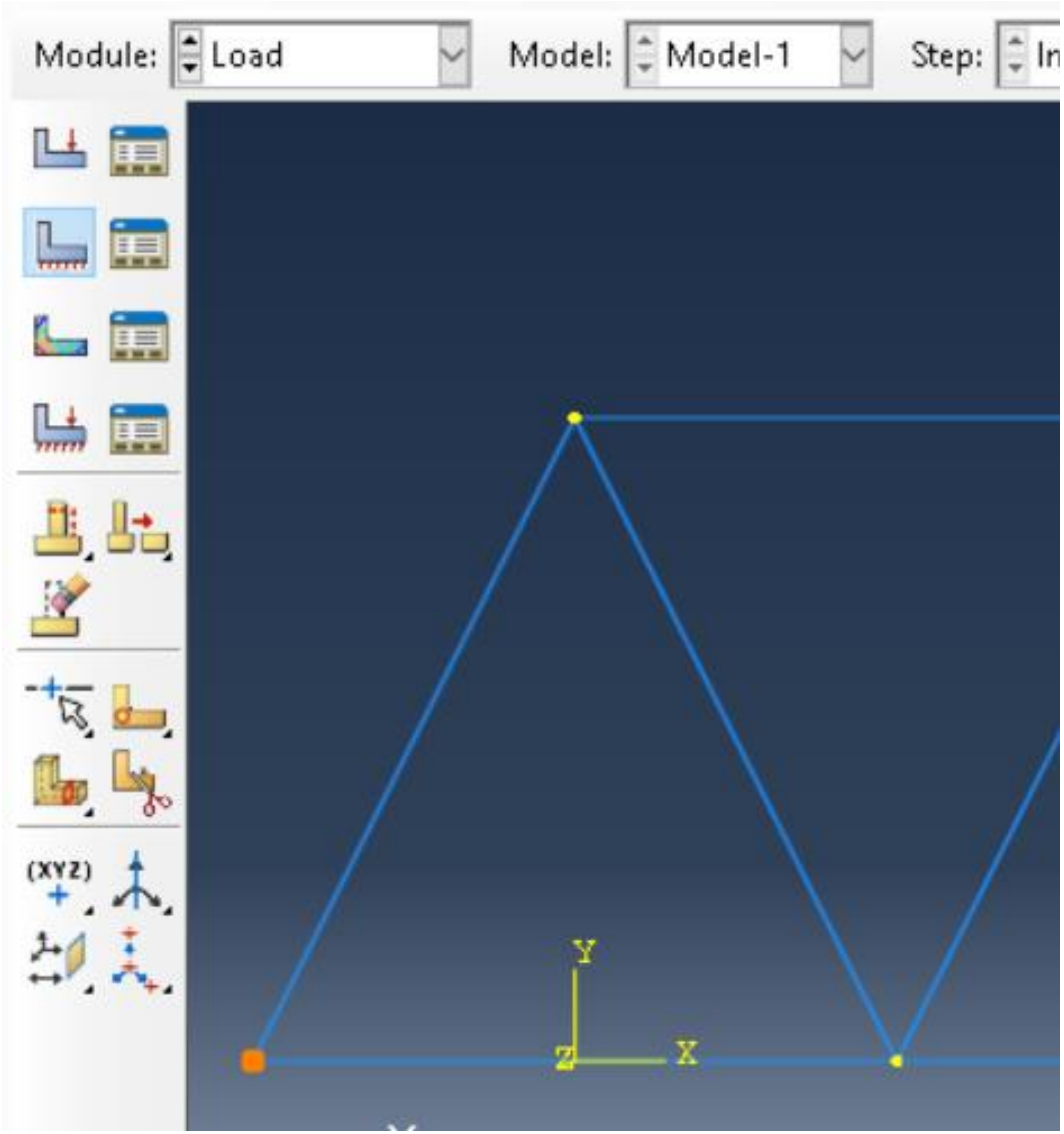



FIGURE 19 Selecting a region to be assigned boundary conditions.

The **Edit Boundary Condition** dialog box appears. Select **PINNED(U1 = U2 = U3 = 0)** and click on **OK**

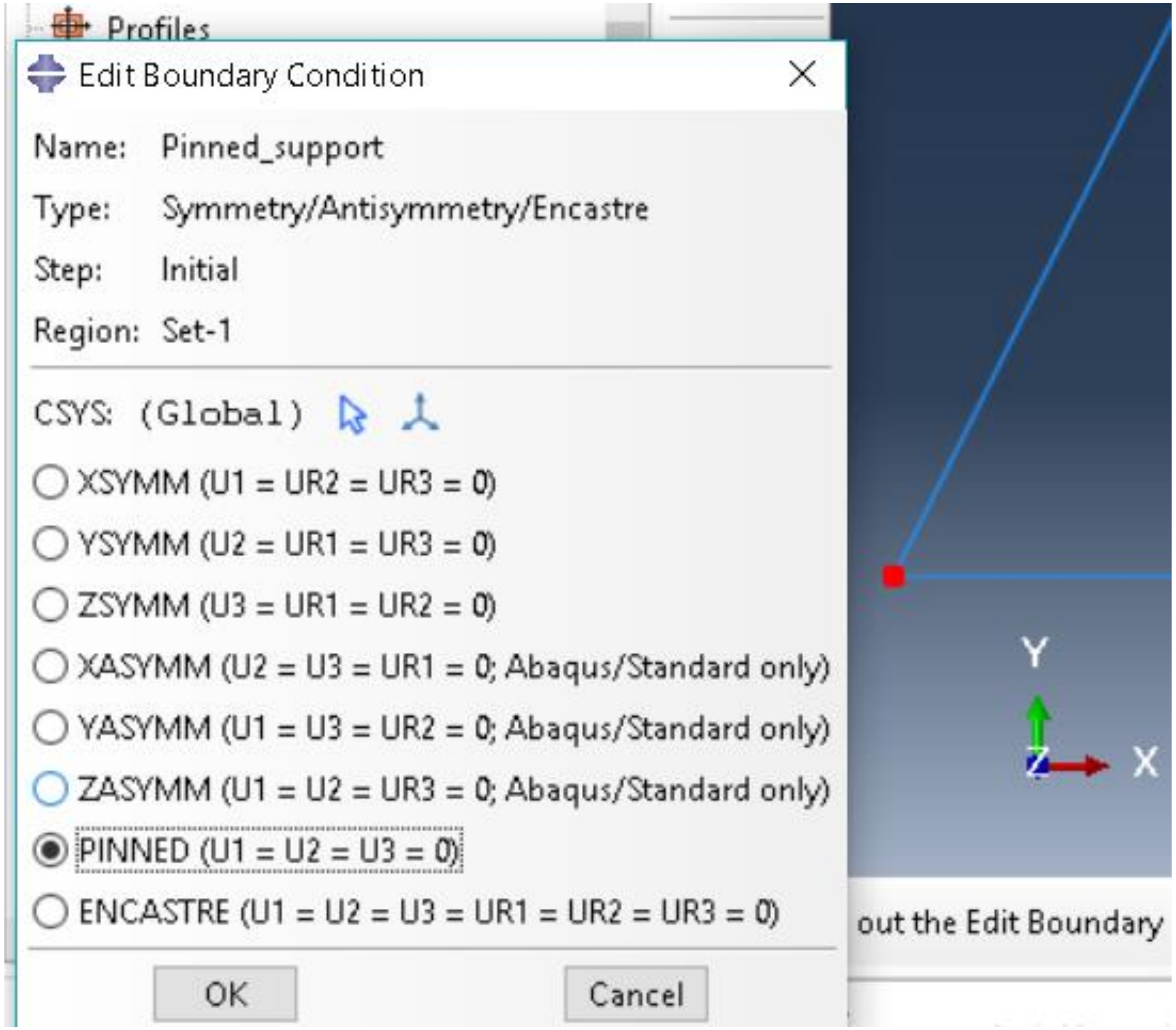



**FIGURE 20** Edit boundary condition dialog box for pinned support.

Under **Steps** and **Initial**, click on **BC** to create the boundary conditions for the roller. In the **Create Boundary Condition** dialog box, name the boundary condition **Roller_Support**. Choose **Symmetry/Antisymmetry/Encastré** and click on **Continue**. Select the right support and click on **Done**. In the **Edit Boundary Condition** dialog box, select **XASYMM(U2 = U3 = UR1 = 0)** and click on **OK**

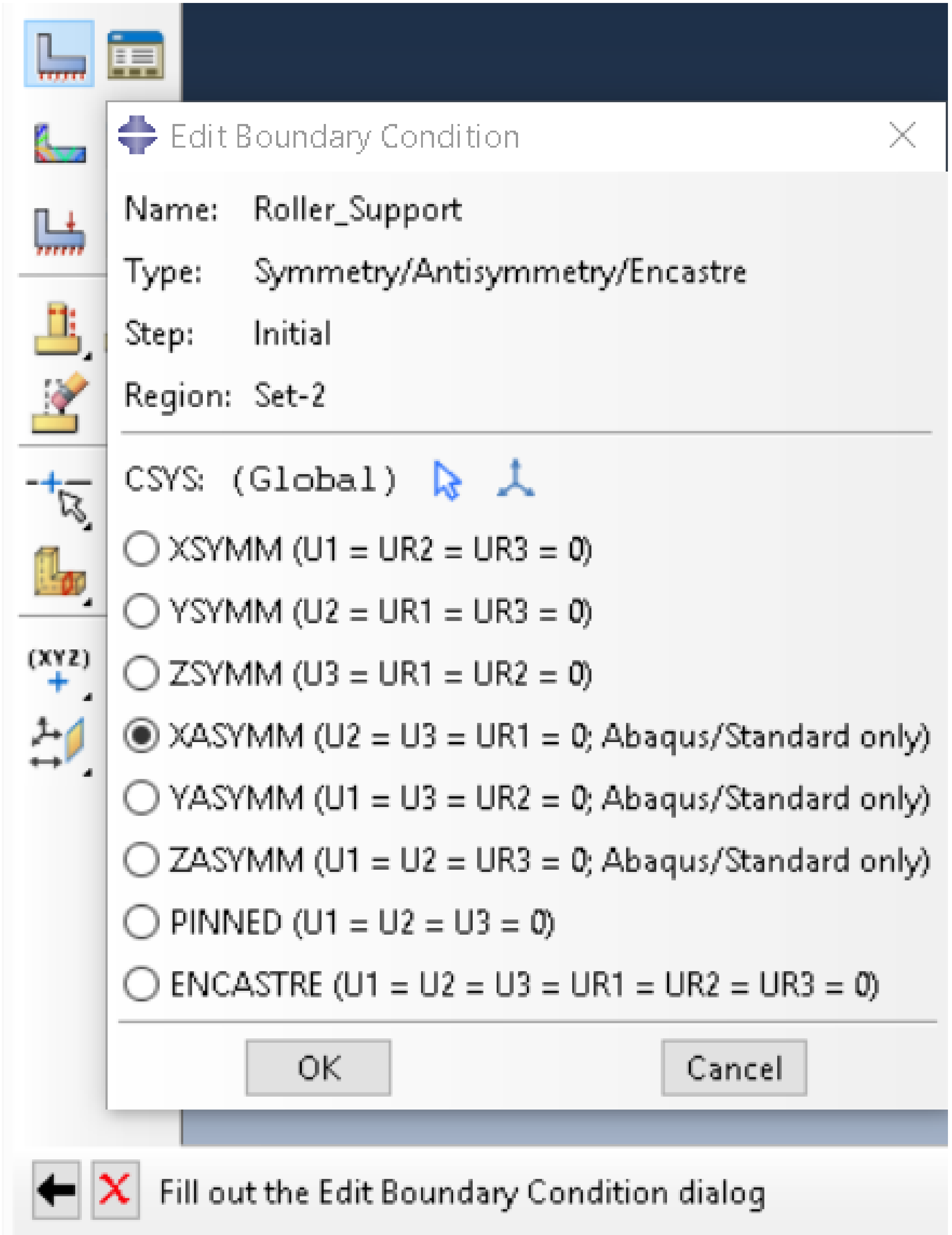



**FIGURE 21** Edit boundary condition dialog box for roller support.

In the left-hand-side menu, right click on **Steps** to crate another step for applying the loads. Click on **Continue**

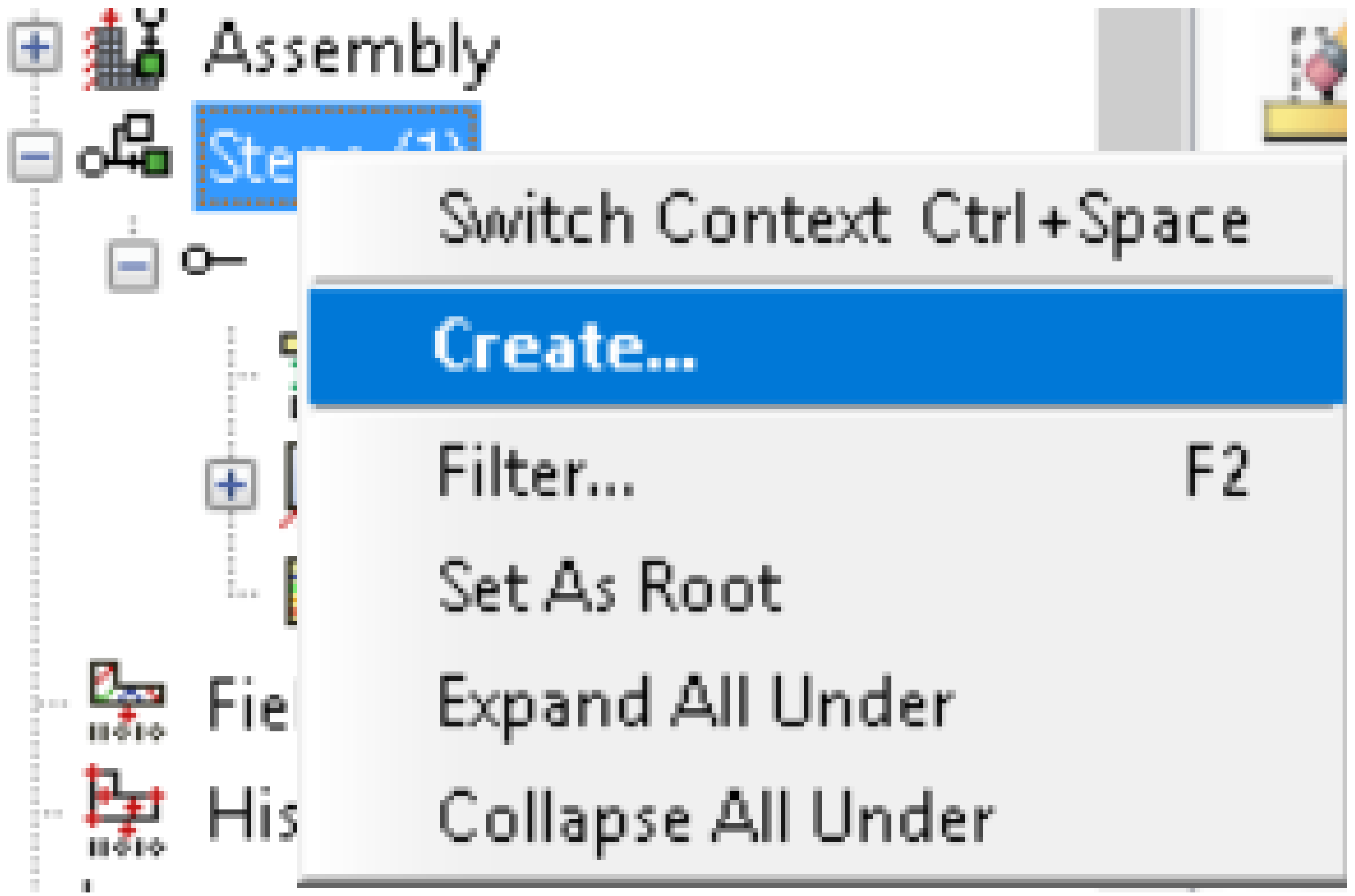



**FIGURE 22** Creating a step for load application.

In the **Create Step** dialog box, name the step **Apply_Loads**, select **Static, General**, and click on **Continue**

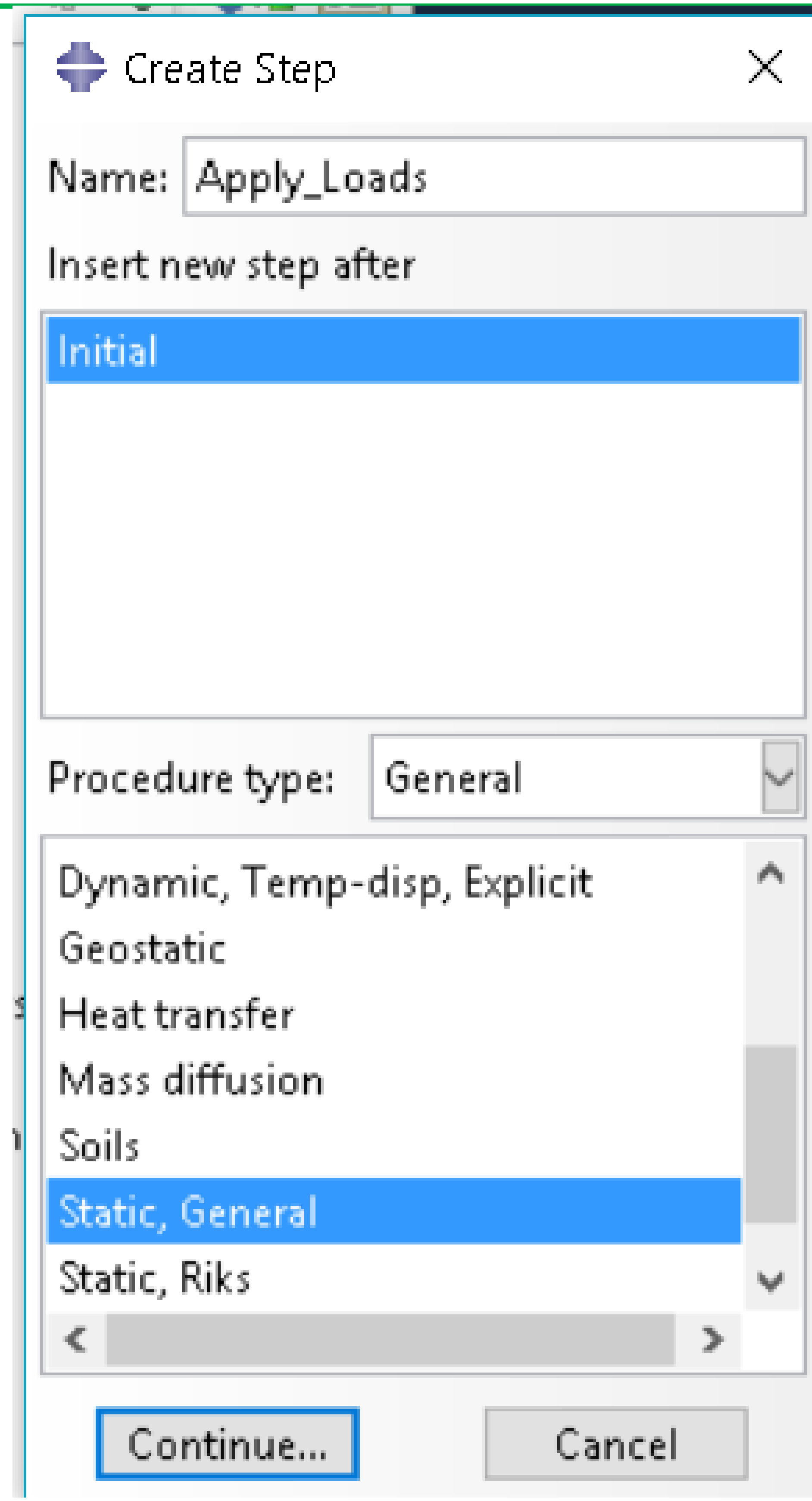



FIGURE 23 Create step dialog box.

In the **Edit step** dialog box, although it is not necessary, you can still provide a description such as applying joint loads. Leave all the other details as they are, and click on **OK**

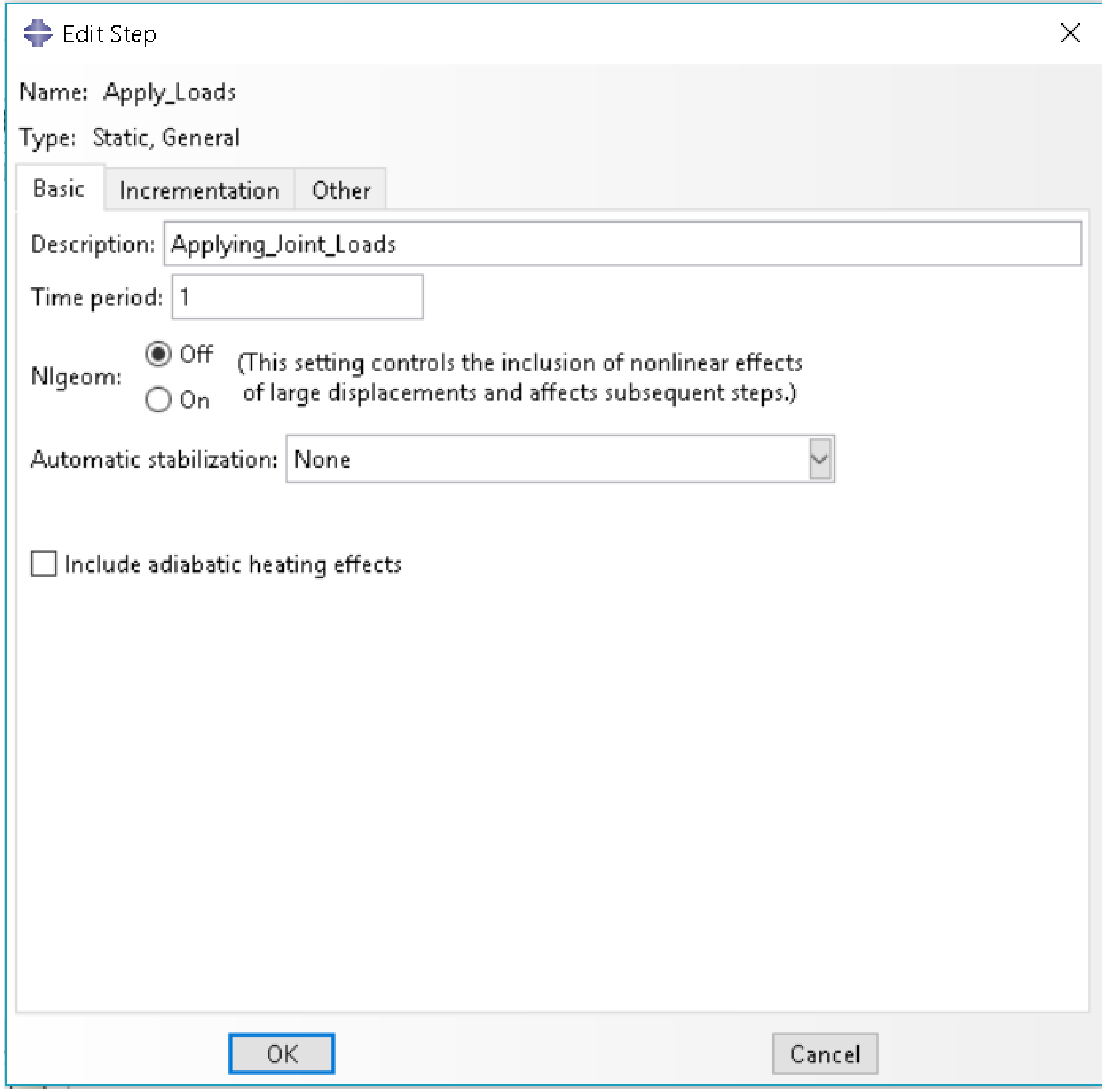



**FIGURE 24** Edit step dialog box.

In the left-hand-side menu, under **Steps** and **Apply_Loads**, click on **Loads** as shown in Figure

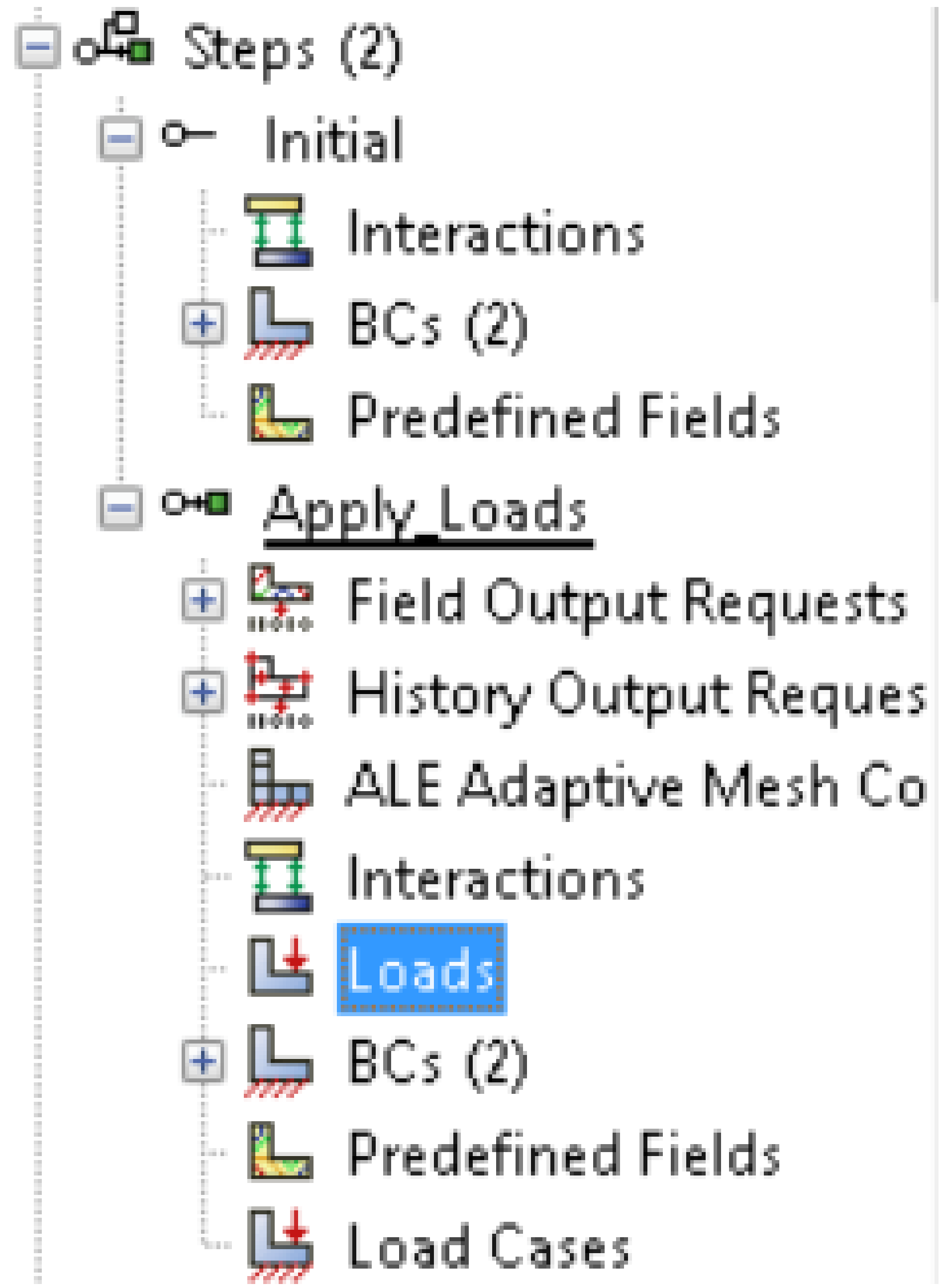



**FIGURE 25** Creating a load.

In the **Create load** dialog box, name the load *Horizontal* 15 kN *force*. In **Step** scroll to **Apply_Loads**, which means that the load will be applied in this step. In **Category** choose **Mechanical**, and in **Type** choose **Concentrated Force**. Click on **Continue**

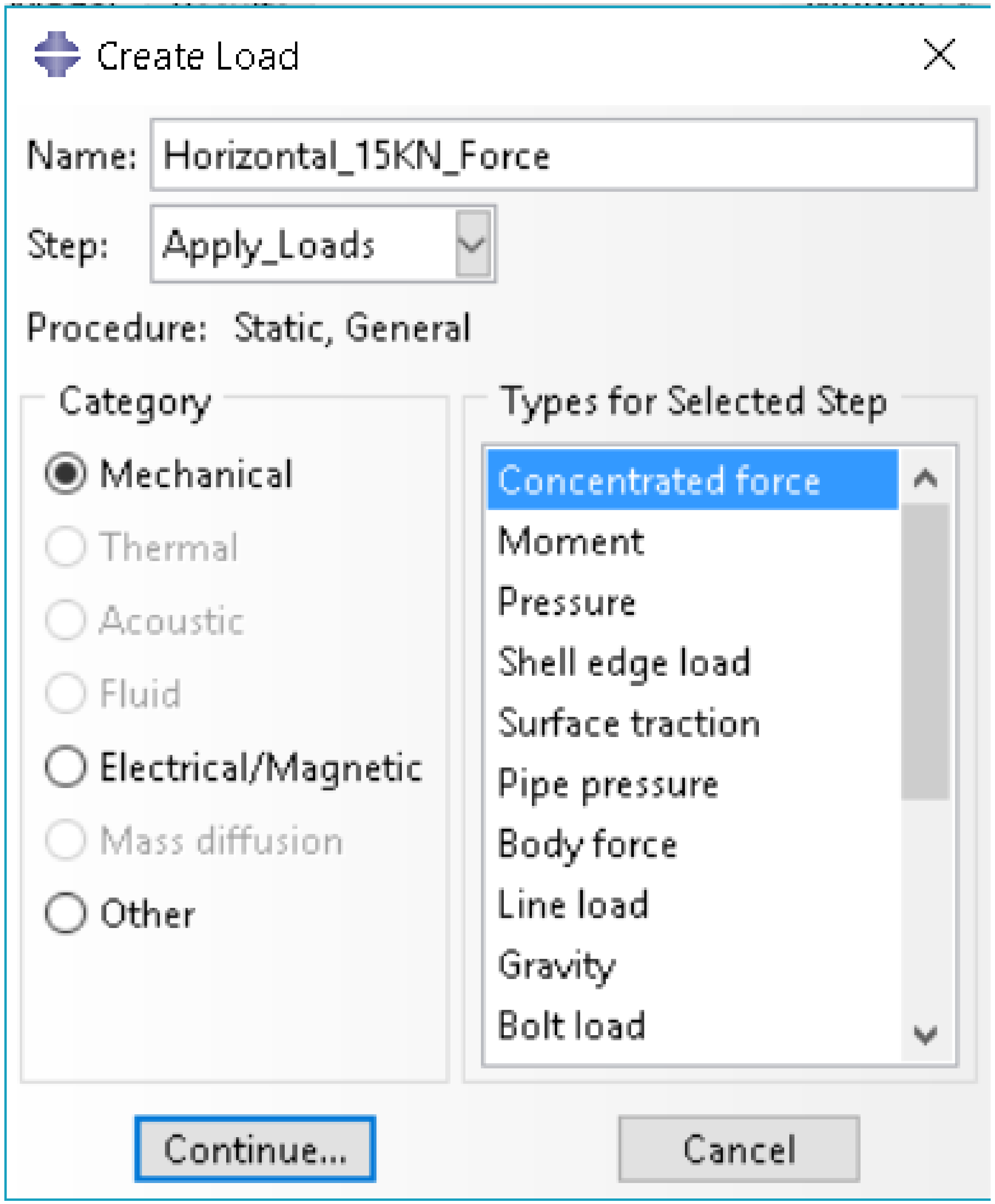



**FIGURE 26** Creating a concentrated load.

With the mouse, select the top-left joint as shown in Figure 27 , and click on done in the bottom-left corner of the same window.

In the **Edit Load** dialog box, enter 15. for **CF1**, and click on **OK**

Repeat the same procedure for the other joint loads. Since they are vertical loads pointing in opposite direction to the axis *y*, their magnitude should be entered in **CF2** as negative. Once finished, the loaded truss should look like the one shown in Figure.

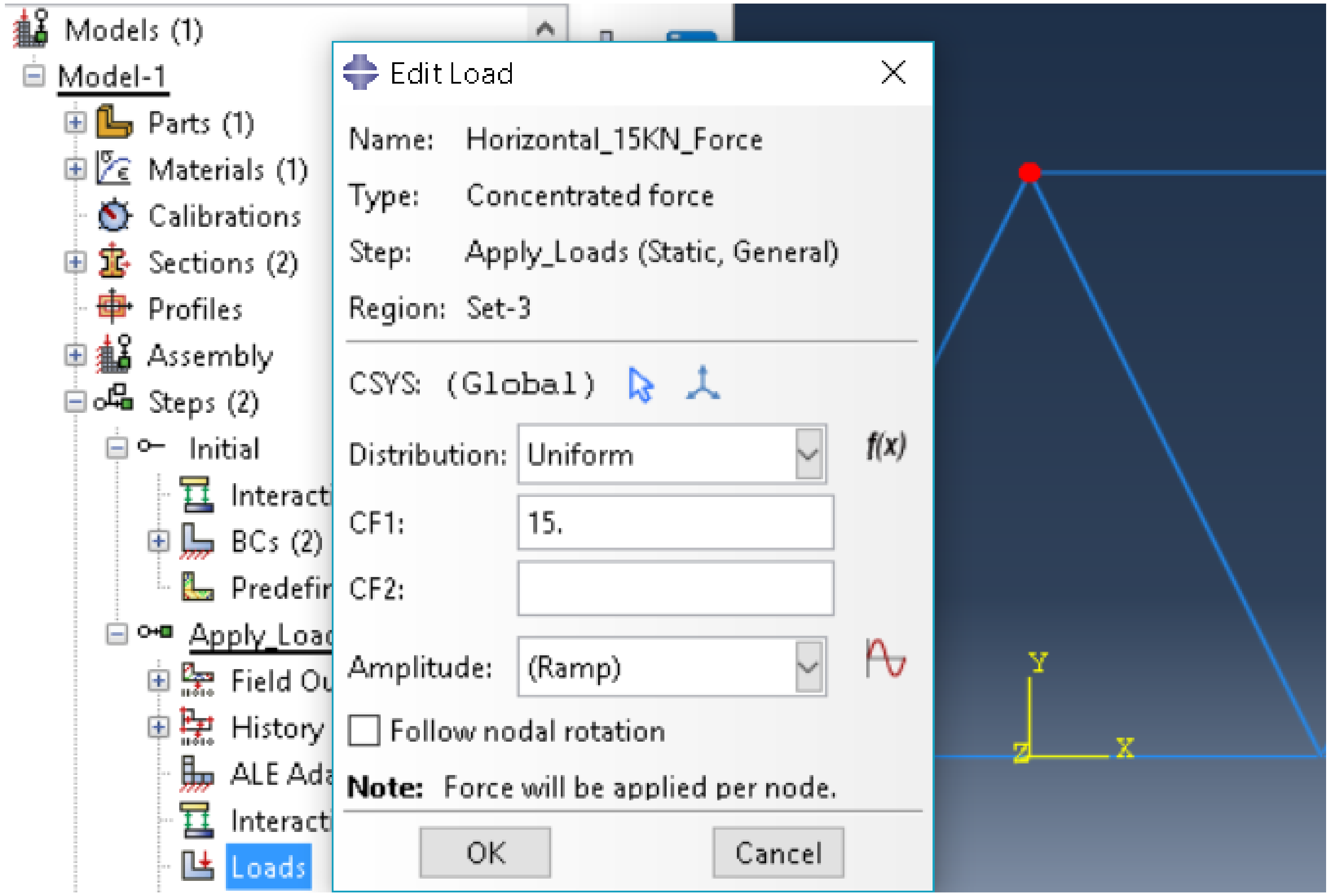
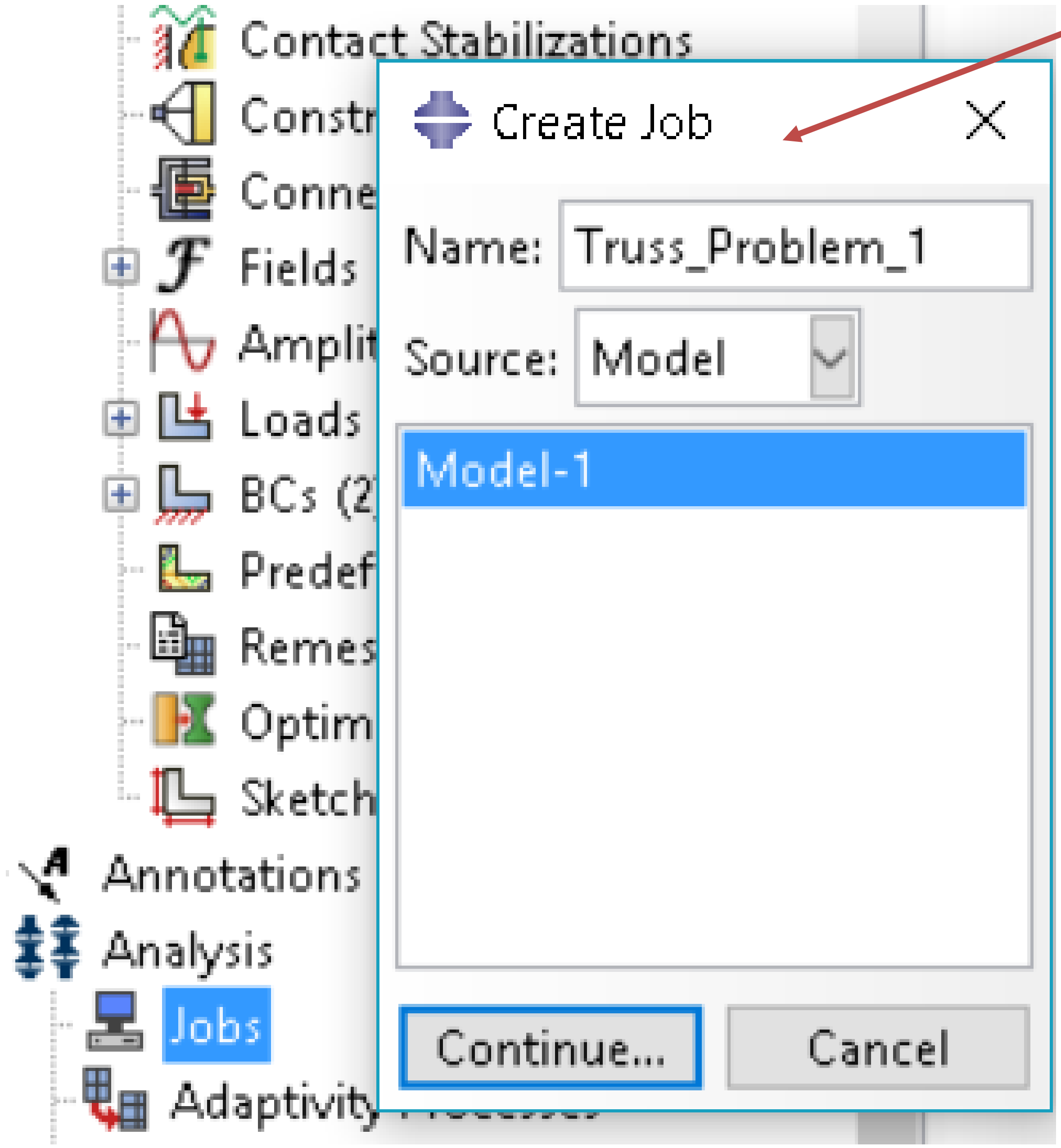



**FIGURE 27** Selecting a joint for load application.

Under **Analysis**, right click on **Jobs** and then click on **Create**

The **Create Job** dialog box appears. Name the job **Truss_Problem_1**, and click on **Continue**



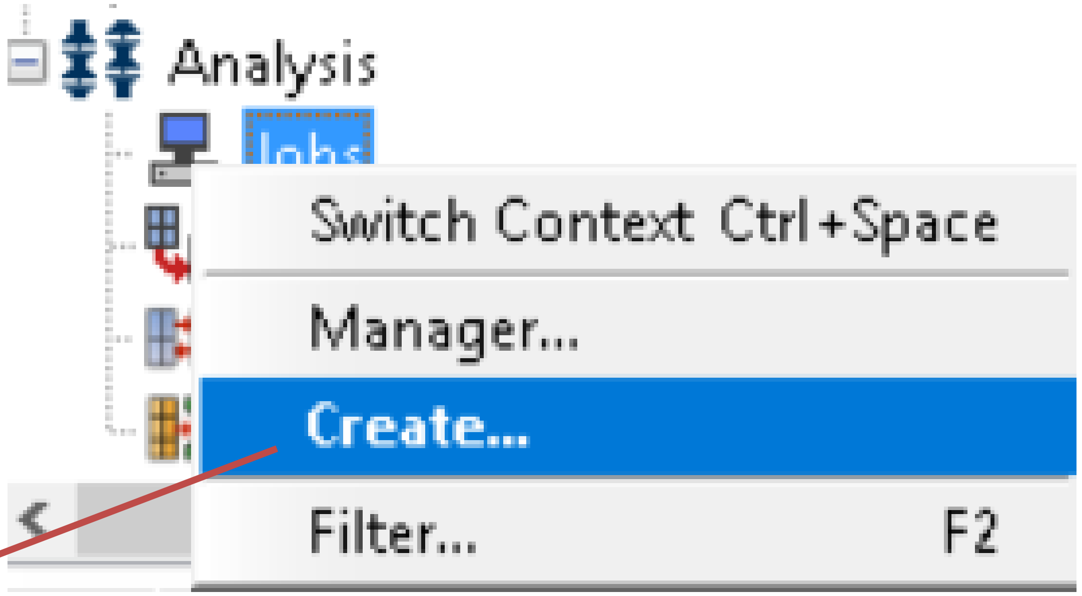




FIGURE 28 Creating a job.

The **Edit Job** dialog box appears. Enter a description for the job. Check **Full analysis** and choose to run the job in **Background** and check to start it **immediately**. Click **OK**

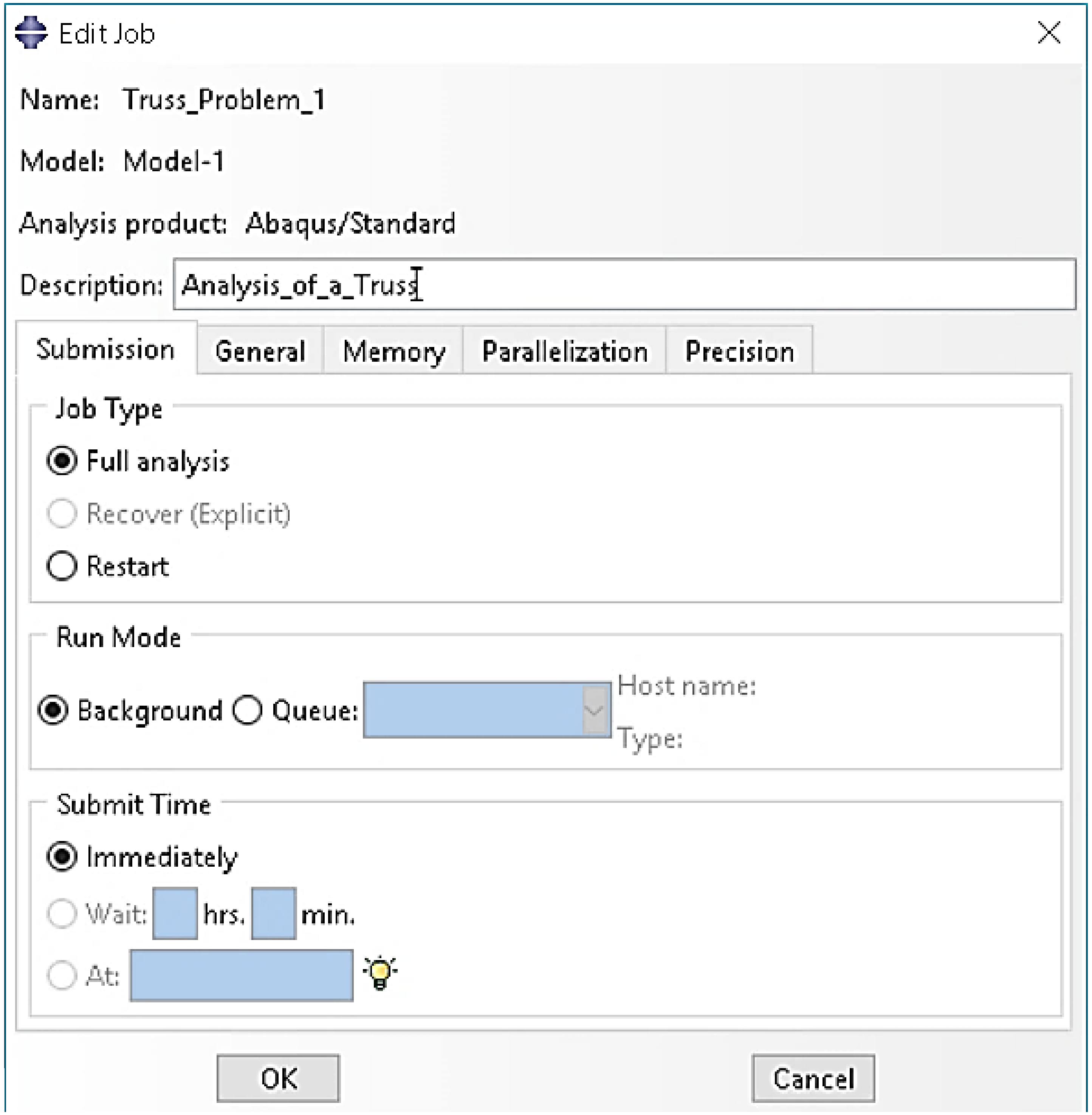



FIGURE29 Editing a job.

Expand the tree under **Jobs**, right click on **Truss_Problem_1**. Then, click on **Submit**

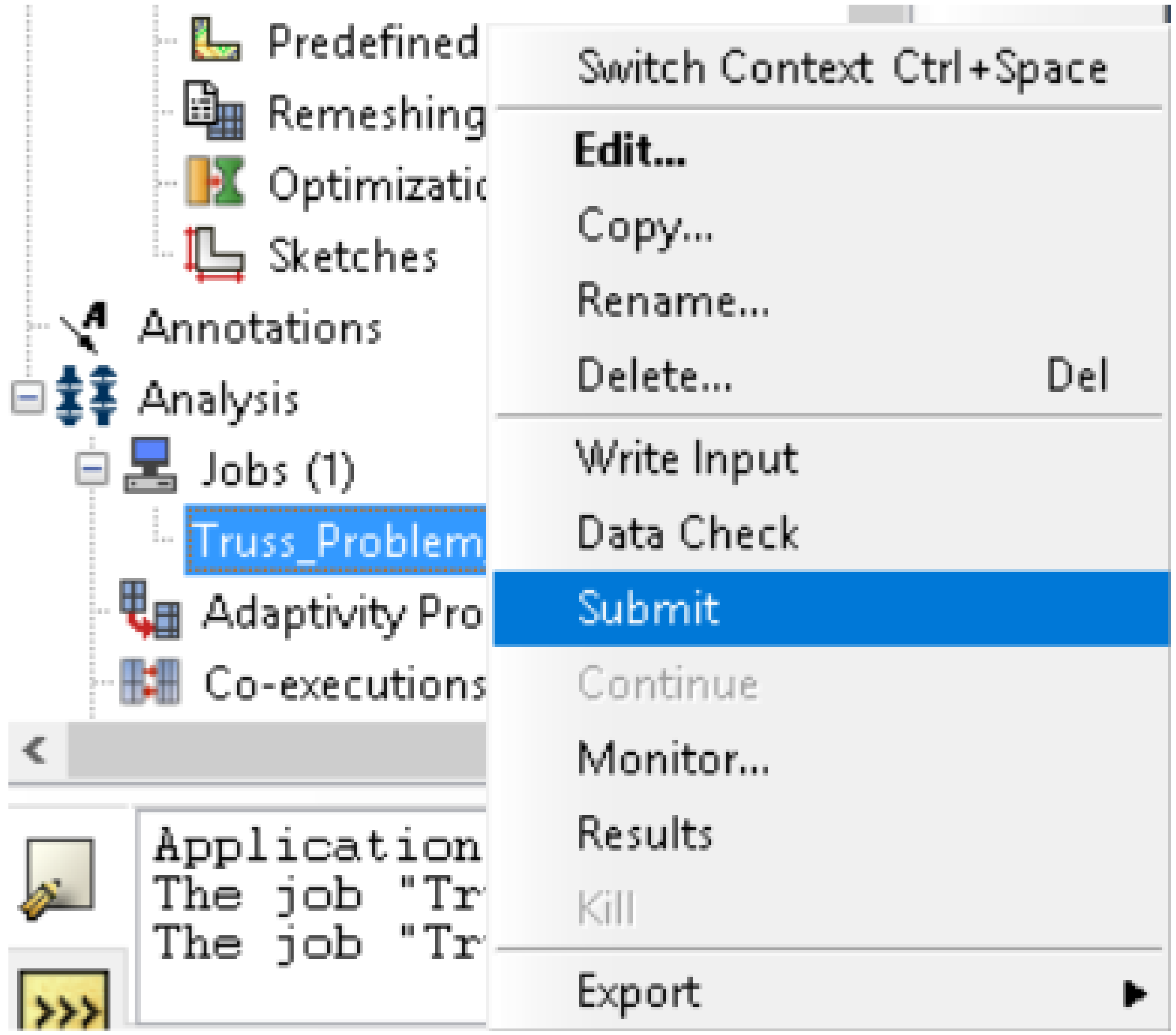



**FIGURE 30** Submitting a job.

If you get the following message **Job Truss_Problem_1 completed successfully** in the bottom window, then your job is free of errors and was executed properly. Now, it is time to view the analysis results

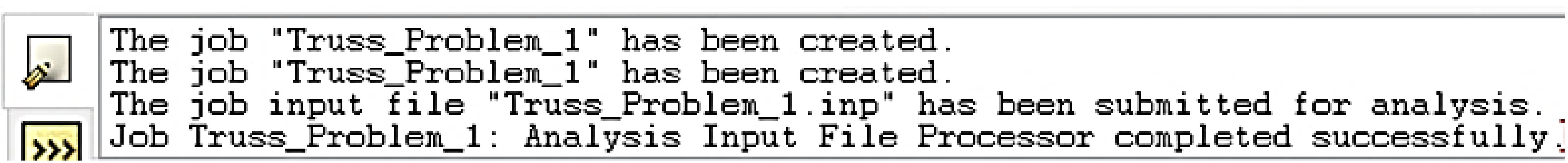



**FIGURE 31** Monitoring of a job.