# Chapter 2
# Rods and Trusses

## 2.1 Definition of Rod Elements

The definition of rod elements is summarized in Table 2.1. The derivation in lectures normally starts with the introduction of an elemental coordinate system ($x$) which is aligned with the principal axis of the element. Based on the definition of this element, deformations ($u_{1x}, u_{2x}$) can only occur along the principal axis. Assuming linear interpolation functions for the displacements, a constant elemental stress and strain is obtained.

The implementation of a rod element in a commercial finite element code is more general, i.e. based on the global coordinate system ($X, Y, Z$). Thus, the geometry is defined based on the global coordinates of each node ($X_i, Y_i, Z_i$) and the cross-sectional area $A$. In such a configuration, each node has three degrees of freedom, i.e. the three displacements expressed in the global coordinate system: $u_{iX}, u_{iY}, u_{iZ}$. Nevertheless, the stress and strain are uniaxial in the truss member. In the case of the linear straight truss (MSC Marc element type 9), the stiffness matrix is obtained based on an one-point integration rule whereas the mass matrix is obtained based on a two-point integration rule.

## 2.2 Basic Examples

### *2.2.1 1D Rod—Fixed Displacement*

#### Problem Description

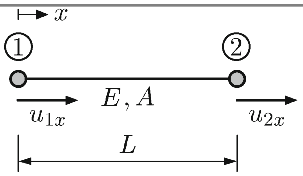
Given is a rod of length $L = 1.0$ and constant axial tensile stiffness given by $E = 20$ and an area $A = 0.5$ as shown in Fig. 2.1. At the left-hand side there is a fixed support and at the right-hand side there is a prescribed displacement of $u_0 = 0.5$. Discretize the problem with a single rod element (MSC Marc element type 9) and calculate the



A. Öchsner and M. Öchsner, *A First Introduction to the Finite Element Analysis Program MSC Marc/Mentat*, https://doi.org/10.1007/978-3-319-71915-3_2

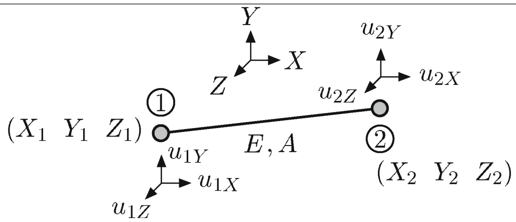
**Table 2.1** Definition of rod elements

Simplified definition (derivation in lecture)




| Definitions | Degrees of freedom |
|---|---|
| Material: $E$ | $u_{1x}, u_{2x}$ |
| Geometry: $L$, $A$ | |

General definition (MSC Marc element type 9)




| Definitions | Degrees of freedom |
|---|---|
| Material: $E$ | Node 1: $u_{1X}, u_{1Y}, u_{1Z}$ |
| Geometry: $A$ | Node 2: $u_{2X}, u_{2Y}, u_{2Z}$ |
| Node 1: $X_1$, $Y_1$, $Z_1$ | |
| Node 2: $X_2$, $Y_2$, $Z_2$ | |

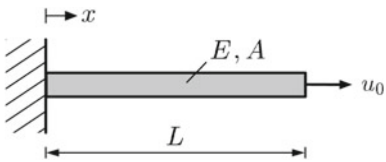



**Fig. 2.1** Schematic drawing of a single rod loaded by an end displacement $u_0$

reaction force at the right-hand node.

## Marc Solution

Under File → Save As..., save file as 'bar_disp'.

Constructing the mesh (Fig. 2.2)

1. Under Geometry&Mesh : **Basic Manipulation** select Geometry and Mesh (see Fig. 2.3).
2. Under Mesh\Nodes, select Add.
3. *0,0,0* ENTER *1,0,0* ENTER.

4. Under Mesh\Elements, select $\overline{\text{Line (2)}}$ (see Fig. 2.2).
5. Press $\overline{\text{Add}}$.
6. In ⑥, select the two nodes with a (LC).
7. Press $\overline{\text{OK}}$.

Setting the Geometric Properties

8. Under Geometric Properties : **Geometric Properties** select New (Structural).
9. Select 3D → Truss (see Fig. 2.4).
10. Set Properties\Area = 0.5.
11. Under Entities\Elements, press $\overline{\text{Add}}$.
12. In ⑥ select the element with a (LC), then (RC).
13. Press $\overline{\text{OK}}$.

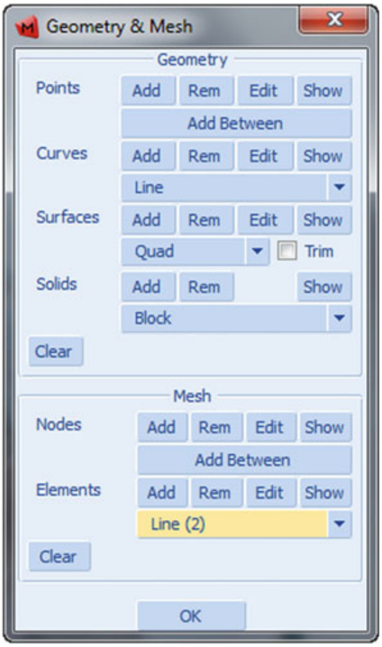



**Fig. 2.2** Geometry and mesh dialog window


**Fig. 2.3** Geometry and mesh

Setting the Material Properties

14. Under Material Properties : **Material Properties** select New → Finite Stiffness Region → Standard (see Fig. 2.5).

15. Set Other Properties\Young's Modulus = 20 (see Fig. 2.6).

16. Under Entities\Elements, press Add.

17. In ⑥ select the element with a (LC), then (RC).

18. Press OK.

Setting the Boundary Conditions

Fixed Support

19. Under Boundary Conditions : **Boundary Conditions** select New (Structural) → Fixed Displacement (see Fig. 2.7).

20. Under Properties tick Displacement X, Displacement Y and Displacement Z.

21. Under Entities\Nodes, press Add.

22. Under ⑥ select the left most node *(0,0,0)* with a (LC), then (RC).

23. Press OK.

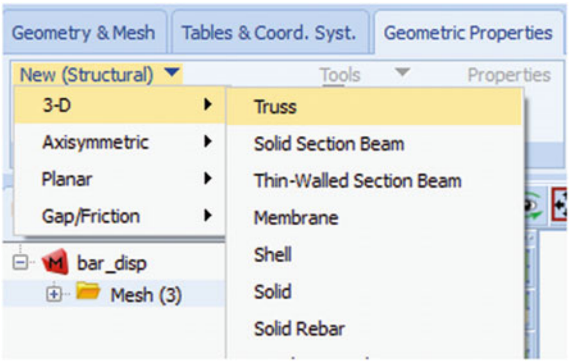



**Fig. 2.4** Geometric properties

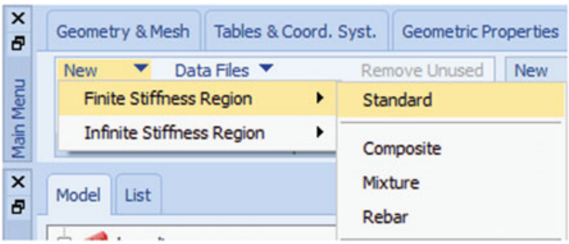



**Fig. 2.5** Material properties selection

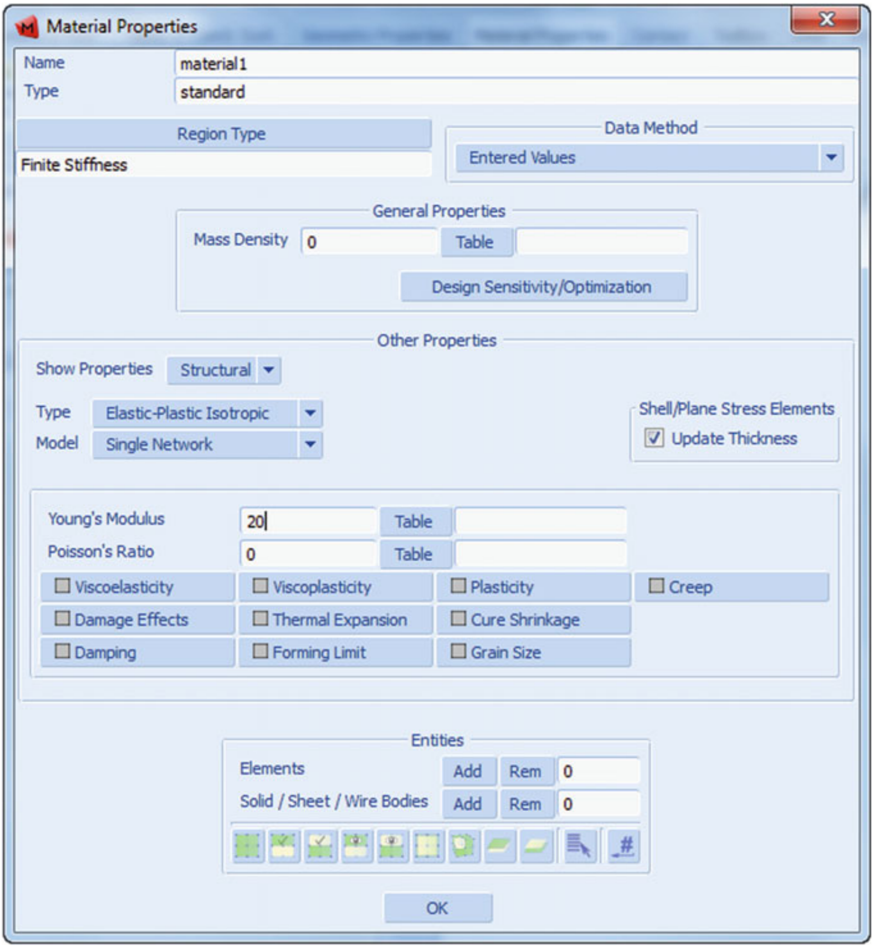



**Fig. 2.6** Material properties definition

Displacement Boundary Condition

24. Under Boundary Conditions: **Boundary Conditions** select New (Structural) → Fixed Displacement. Set Name = disp1.

25. Under Properties tick Displacement X. Set Displacement X = 0.5 (see Fig. 2.8).

26. Under Entities\Nodes, press Add.

27. Under ⑥ select the right most node *(1,0,0)* with a (LC), then (RC).

28. Press OK.

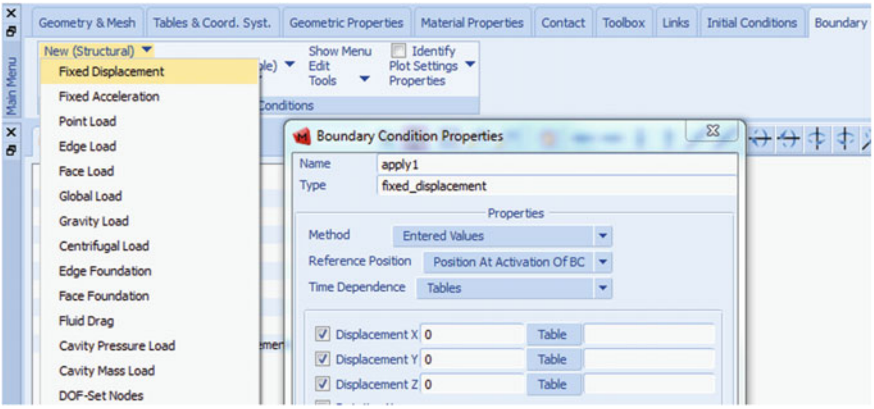
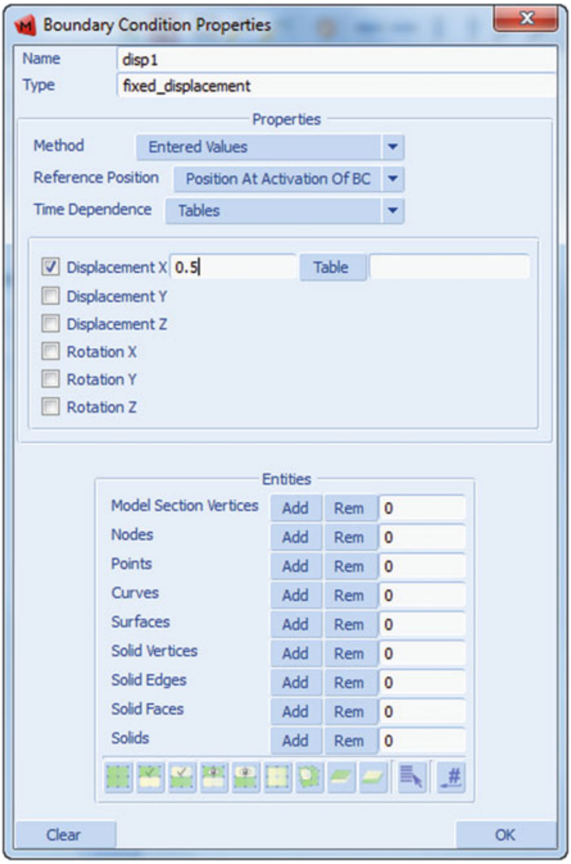



**Fig. 2.7** Boundary conditions dialog window

**Fig. 2.8** Entering the displacement boundary condition



Running the Job

29. Under Jobs : **Jobs** select New → Structural.
30. Press Check; See in ⑧ if there are any errors.
31. If there are none, press Run.
32. In "Run Job", press Advanced Job Submission (see Fig. 2.9).
33. Press Save Model.
34. Press Write Input File. Press OK.
35. Press Submit 1.
36. Wait until Status = Complete.
37. Press Open Post File (Model Plot Results Menu).

Viewing the model

38. Under Deformed Shape\Style, select Deformed and Original.

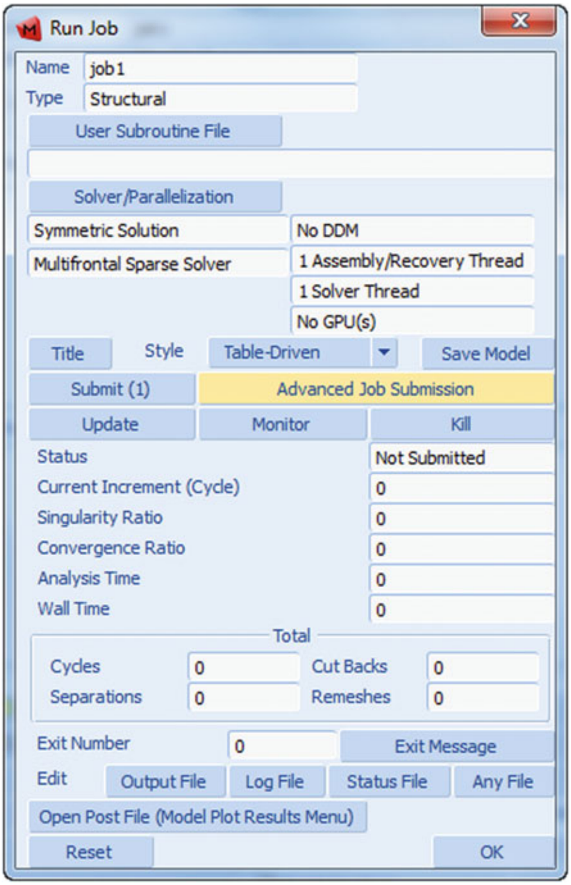
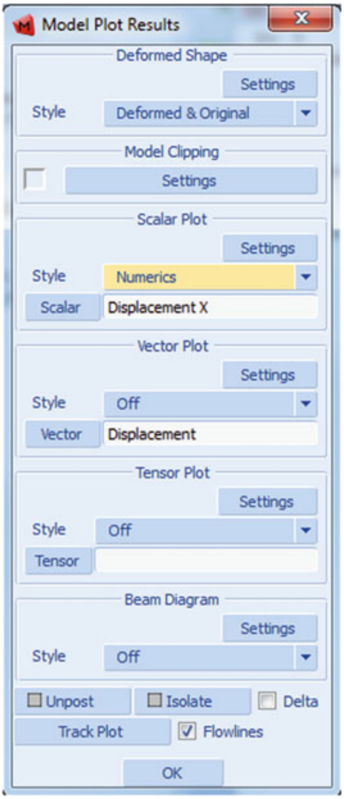



**Fig. 2.9** Advanced job submission

**Fig. 2.10** Job result dialog window




39. Under Scalar Plot\Style, select Numerics (see Fig. 2.10).
40. Under Scalar Plot, press Scalar and select Displacement X.
41. Under Scalar Plot, press Scalar and select Reaction Force X.
42. Press OK.

### Result

The reaction force at the right-hand end of the rod is found to be 5.

### Additional Questions

1. Determine the stress and strain inside the element (via nodal values).
2. Calculate the analytical solution for stress and strain based on HOOKE's law.
3. Repeat the problem based on two elements of equal length ($L' = 0.5$) and determine the displacement, stress and strain at the middle and right-hand node.
4. Perform a finite element 'hand calculation' to determine the displacements (as a function of the given variables $u_0$, $E$, $A$, $L'$) in the middle and the right-hand end for the two-element problem.

### 2.2.2 1D Rod—Fixed Point Load

#### Problem Description

Given is a rod of length $L = 1.0$ with a constant axial tensile stiffness given by $E = 20$ and $A = 0.5$ as shown in Fig. 2.11. At the left-hand side there is a fixed support and the right-hand side is loaded by a single force $F_0 = 5$. Use a single rod element to determine the elongation of the right-hand end.

#### Marc Solution

The steps for this example are the same as the one of the previous example, Sect. 2.2.1 except for steps 24–28 and the file name, which should be 'bar_force' (see Sect. 2.2.1). The following steps have to replace these:

Setting Fixed Point Load
24. Under Boundary Conditions : **Boundary Conditions** select New (Structural) → Point Load. Set Name = force1.
25. Under Properties, tick Force X. Set at 5.
26. Under Entities\Nodes, press Add (see Fig. 2.12).
27. Under ⑥ select the right most node (*1,0,0*) with a (LC), then (RC).
28. Press OK.

#### Result

The resulting displacement on the right-hand end of the rod is found to be 0.5.

#### Additional Questions

1. Determine the stress and strain inside the element (via nodal values).
2. Calculate the analytical solution for stress and strain based on HOOKE's law.
3. Repeat the problem based on two elements of equal length ($L' = 0.5$) and determine the displacement, stress and strain at the middle and right-hand node.
4. Perform a finite element 'hand calculation' to determine the displacements (as a function of the given variables $u_0$, $E$, $A$, $L'$) in the middle and the right-hand end for the two-element problem.

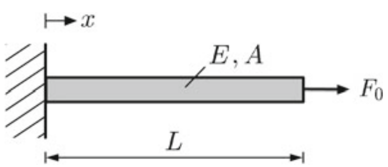



**Fig. 2.11** Schematic drawing of a single rod loaded by an end load $F_0$

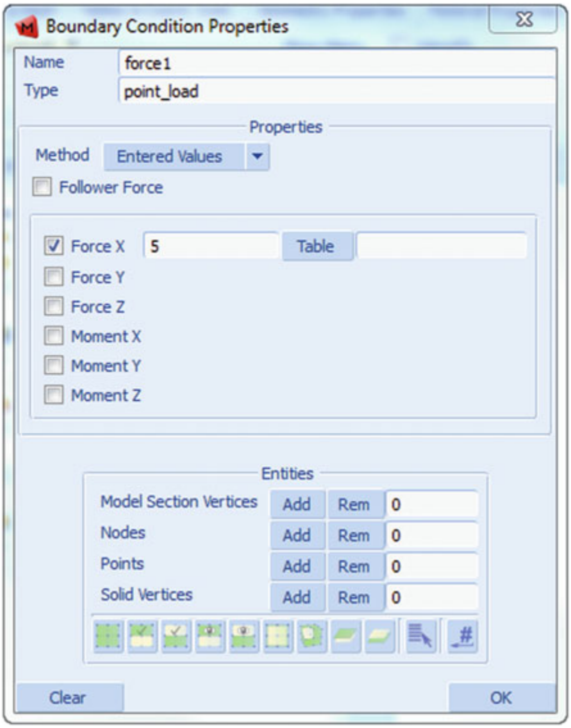



**Fig. 2.12** Entering fixed point load value

### 2.2.3 *1D Rod—Multiple Loadcases*

#### Problem Description

Given is a rod of length $L = 1.0$ with a constant axial tensile stiffness given by $E = 20$ and $A = 0.5$ as shown in Fig. 2.13. At the left-hand side there is a fixed support and the right-hand side is

(I) elongated by a given displacement $u_0 = 0.5$, or,
(II) loaded by a single force $F_0 = 5$.

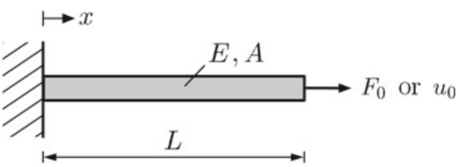



**Fig. 2.13** Schematic drawing of a single rod with different load cases

Discretize the problem with a single rod element to determine:
(a) the reaction force and
(b) the displacement at the right-hand end.

**Marc Solution**
For this example, involving loadcases, the steps are equal to those in example 1, from steps 1–28 (see Sect. 2.2.1). Save as 'bar_twoloads'.
Add the following steps:

Setting the Additional Boundary Condition

Force Boundary Condition

29. Under Boundary Conditions: **Boundary Conditions** select New (Structural) → Point Load. Set Name = force1.
30. Under Properties, tick Force X. Set at 5.
31. Under Entities\Nodes, press Add.
32. Under ⑥ select the right most node *(1,0,0)* with a (LC), and confirm with a (RC).
33. Press OK.

Defining Loadcase 1—Fixed Displacement

34. Under Loadcases: **Loadcases** select New → Static.
35. Set Name = fixed_displacement (see Fig. 2.14).
36. Press Loads. Untick force1. Press OK.
37. Set Stepping Procedure\#Steps = 1. Press OK.

Defining Loadcase 2—Fixed Point Load

38. Under Loadcases: **Loadcases** select New → Static.
39. Set Name = fixed_force.
40. Press Loads. Untick displ1. Press OK.
41. Set Stepping Procedure\#Steps = 1 Press OK.

Running the two jobs

42. Under Jobs: **Jobs** select New → Structural. Set Name = force (see Fig. 2.15).
43. Under Available, select fixed_force.
44. Press Initial Loads. Untick displ1. Press OK.
45. Press Check; See in ⑧ if there are any errors.

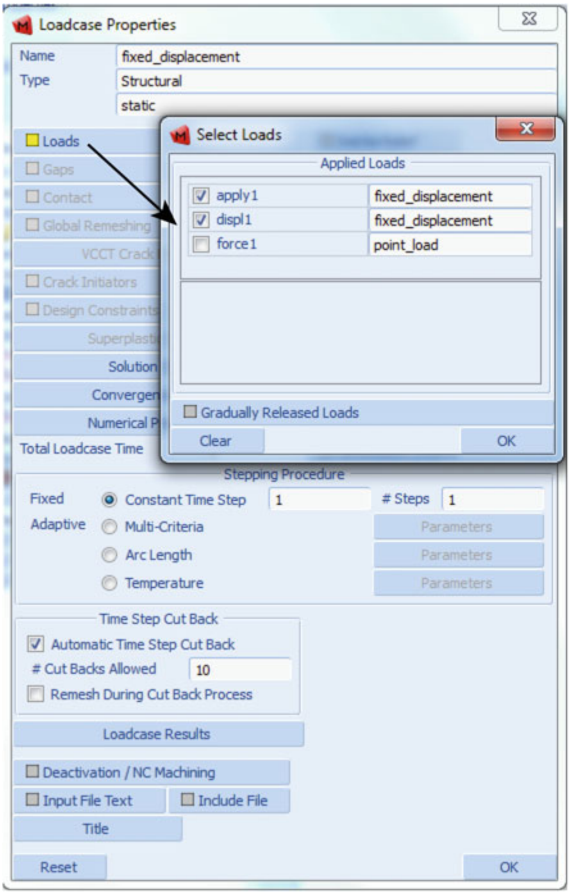



**Fig. 2.14** Defining loadcase 1

46. If there are none, press Run.
    47. In 'Run Job', press Advanced Job Submission.
        48. Press Save Model.
        49. Press Write Input File. Press OK.
        50. Press Submit 1.
    51. Wait until Status = Complete. Press OK.
52. Press OK.

53. Under Jobs: **Jobs** select New → Structural. Set Name = displacement.
    54. Under Available, select fixed_displacement.
    55. Press Initial Loads. Untick force1. Press OK.
    56. Press Check; See in ⑧ if there are any errors.

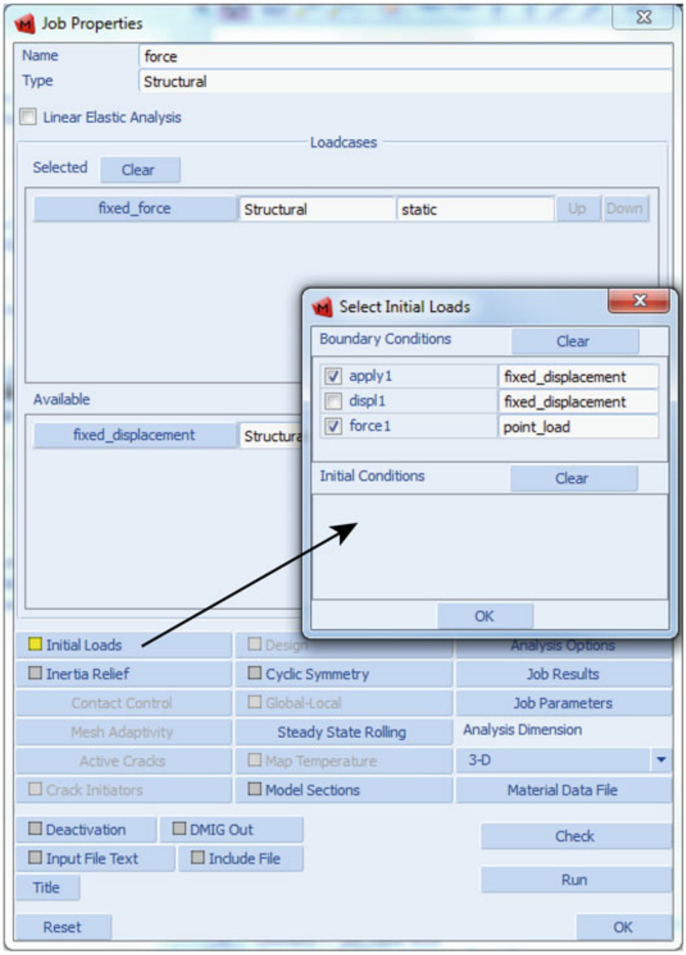



**Fig. 2.15** Running Job 1

57. If there are none, press Run.
    58. In 'Run Job', press Advanced Job Submission.
        59. Press Save Model.
        60. Press Write Input File. Press OK.
        61. Press Submit 1.
    62. Wait until Status = Complete.
    63. Press Open Post File (Model Plot Results Menu).

Viewing the model

64. Under Deformed Shape\Style, select Deformed and Original.
65. Under Scalar Plot\Style, select Numerics.
66. Press Scalar and select Reaction Force X. Press OK.
67. Press OK.

Open Job fixed_force

68. Under File → Results → Open. Open bar_twoloads_force.t16 (see Fig. 2.16).
69. In the window 'Model Plot Results', under Scalar Plot, press Scalar and select Displacement X.
70. Press OK.

**Results**

Loadcase I: The reaction force on the right-hand end of the rod, is found to be 5.
Loadcase II: The resulting displacement on the right-hand end of the rod is found to be 0.5.

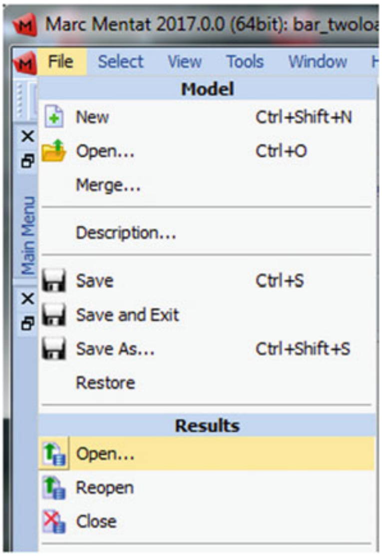



**Fig. 2.16** Opening fixed_force result case

### 2.2.4 *Plane Truss—Triangle*

#### Problem Description

Given is the two-dimensional truss structure as shown in Fig. 2.17 where the trusses are arranged in the form of an equilateral triangle (internal angles $\beta = 60°$). The three trusses have the same length $L = 1.0$, the same YOUNG's modulus $E = 20$ and the same cross-sectional area $A = 0.5$. The structure is loaded by
(I) a horizontal force $F = 0.1$ at node 2, or,
(II) a prescribed displacement $u = 0.01$ at node 2.
Determine for (I) the deformation at the load application point and for (II) the reaction force by discretizing the structure with three rod elements.

#### Marc Solution

Save as 'bar_triang'

Constructing the mesh

1. Under Geometry&Mesh : **Basic Manipulation** select Geometry and Mesh.
2. Under Mesh\Nodes, select Add.
3. *0,0,0* ENTER *0,1,0* ENTER *−0.866,0.5,0* ENTER.
4. Under Mesh\Elements, select Line (2).
5. Press Add.
6. In ⑥ (RC) on node 1 and 2, node 2 and 3, and node 3 and 1.
7. Press OK.

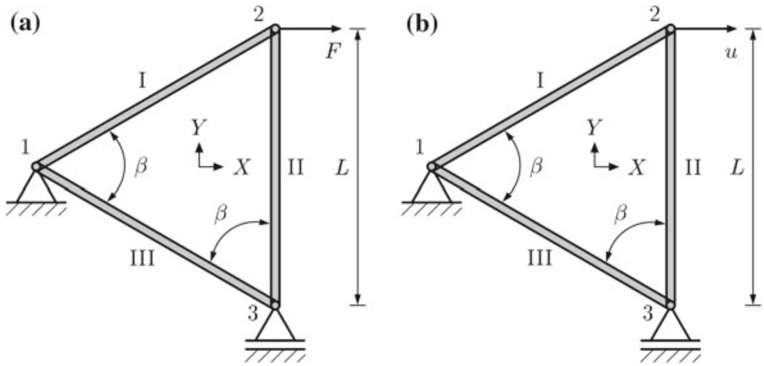



**Fig. 2.17** Plane truss triangle structure: **a** force boundary condition; **b** displacement ($u \neq 0$) boundary condition

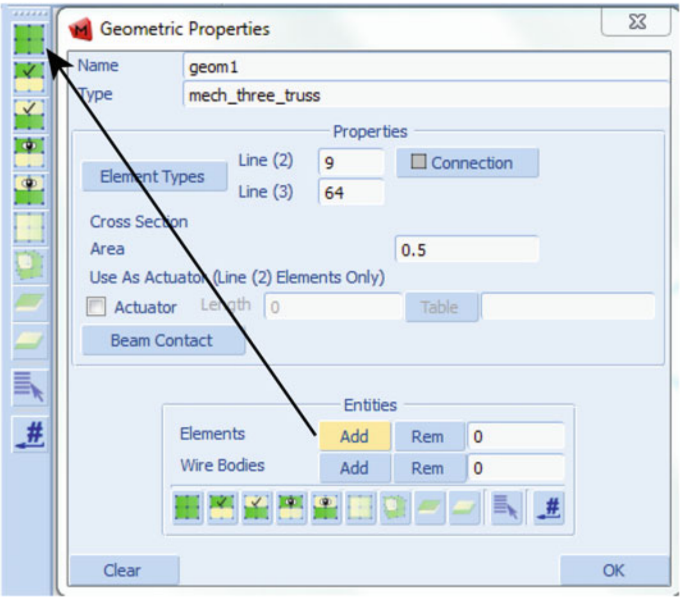



**Fig. 2.18** Selecting all elements

Setting the Geometric Properties

8. Under [Geometric Properties]: **Geometric Properties** select New (Structural).
9. Select 3D → Truss (see Fig. 2.4).
10. Set Properties\Area = 0.5 (see Fig. 2.18).
11. Under Entities\Elements, press Add.
12. In ⑦ (see Fig. 1.4), press the first symbol 'All Existing'.
13. Press OK.

Setting the Material Properties

14. Under [Material Properties]: **Material Properties** select New → Finite Stiffness → Standard.
15. Set Other Properties\Young's Modulus = 20 (See Fig. 2.6).
16. Under Entities\Elements, press Add.
17. In ⑦, press the first symbol 'All Existing'.
18. Press OK.

Setting the Boundary Conditions

Fixed Support

19. Under Boundary Conditions: **Boundary Conditions** select New (Structural) → Fixed Displacement (see Fig. 2.7).
20. Under Properties tick Displacement X and Displacement Y.
21. Under Entities\Nodes, press Add.
22. Under ⑥ select the left most node (*-0.866,0.5,0*) with a (LC), then (RC).
23. Press OK.

24. Under Boundary Conditions: **Boundary Conditions** select New (Structural) → Fixed Displacement.
25. Under Properties tick Y.
26. Under Entities\Nodes, press Add.
27. Under ⑥ select the bottom right node (*0,0,0*) with a (LC), then (RC).
28. Press OK.

Fixed displacement

29. Under Boundary Conditions: **Boundary Conditions** select New (Structural) → Fixed Displacement. Set Name = displ1.
30. Under Properties tick Displacement X. Set Displacement X = 0.01.
31. Under Entities\Nodes, press Add.
32. Under ⑥ select the top right node (*0,1,0*) with a (LC), then (RC).
33. Press OK.

Point load

34. Under Boundary Conditions: **Boundary Conditions** select New (Structural) → Point Load. Set Name = force1.
35. Under Properties, tick Force X. Set at 0.1.
36. Under Entities\Nodes, press Add.
37. Under ⑥ select the top right node (*0,1,0*) with a (LC), then (RC).
38. Press OK.

Then follow steps 34 onwards from the previous example '1D—Loadcases' (Sect. 2.2.3).

**Results**

The results for the two load cases are obtained as:

(I) $u_X = 0.016$,
(II) Reaction force: $F_X = 0.059$.

**Additional Questions**

1. Determine the stress in each rod element.
2. Write a procedure file to automatically create the nodes and elements.
3. Perform a finite element 'hand calculation' and determine the reactions at the 'load' application point and the stresses in each rod as a function of $E$, $L$, $A$, $F$, and $u$. Simplify these results to the numerical values and compare with the results obtained with MSC Marc.

## 2.3 Advanced Examples

### *2.3.1 Plane Bridge Structure*

**Problem Description**

Given is a simplified plane bridge structure over a valley as shown in Fig. 2.19. The bridge structure is idealized in the $X$-$Y$ plane based on thirteen rod elements (I, ..., XIII) which are connected at eight nodes (1, ..., 8). Consider the following numerical values for the geometrical and material parameters: $L = 4000$ mm; $E = 200000$ MPa; and $A = 10$ mm$^2$. The structure is loaded by:
(I) a vertical force $F_Y = -100$ at node 2,
(II) a vertical force $F_Y = -100$ at node 3.
All elements are rod elements. Determine the deformation $(u_{3X}, u_{3Y})$ at node 3.

**Marc Solution**

Save as 'truss_bridge'

Constructing the mesh

1. Under Geometry&Mesh : **Basic Manipulation** select Geometry and Mesh.
2. Under Mesh\Nodes, select Add. Add nodes 1–8.
3. (1) *0,0,0* Enter (2) *4000,0,0* Enter (3) *8000,0,0* Enter (4) *12000,0,0* Enter (5) *16000,0,0* Enter (6) *12000,4000,0* Enter (7) *8000,4000,0* Enter (8) *4000,4000,0* Enter.
4. Under Mesh\Elements, select Line (2).
5. Press Add.
6. In ⑥ (RC) on the following nodes to form rod elements:
   (1,2),(2,3),(3,4),(4,5),(5,6),(6,7),(7,8),
   (8,1),(8,2),(8,3),(6,3),(7,3),(6,4).
7. Press OK.

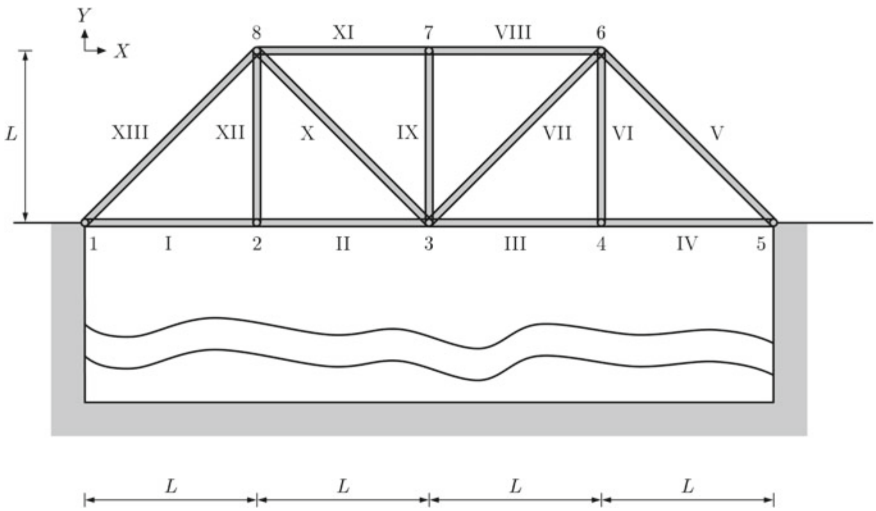



**Fig. 2.19** Plane bridge structure

Setting the Geometric Properties

8. Under Geometric Properties : **Geometric Properties** select New (Structural).
9. Select 3D → Truss (see Fig. 2.4).
10. Set Properties\Area = 10.
11. Under Entities\Elements, press Add.
12. In ⑦, press the first symbol, 'All Existing'.
13. Press OK.

Setting the Material Properties

14. Under Material Properties : **Material Properties** select New → Finite Stiffness → Standard.
15. Set Other Properties\Young's Modulus = 200000 (see Fig. 2.6).
16. Under Entities\Elements, press Add.
17. In ⑦, press the first symbol, 'All Existing'.
18. Press OK.

Setting the Boundary Conditions

Support Conditions

19. Under Boundary Conditions : **Boundary Conditions** select New (Structural) → Fixed Displacement (see Fig. 2.7).
20. Under Properties tick Displacement X and Displacement Y.

21. Under Entities\Nodes, press Add.
22. Under ⑥ select the left and right most node (*0,0,0* and *16000,0,0*) with a (LC), then (RC).
23. Press OK.

Point Load 1

24. Under Boundary Conditions: **Boundary Conditions** select New (Structural) → Point Load. Set Name = force_node_2.
25. Under Properties, tick Force Y. Set at −100.
26. Under Entities\Nodes, press Add.
27. Under ⑥ select node 2 (*4000,0,0*) with a (LC), then (RC).
28. Press OK.

Point Load 2

29. Under Boundary Conditions: **Boundary Conditions** select New (Structural) → Point Load. Set Name = force_node_3.
30. Under Properties, tick Force Y. Set at −100.
31. Under Entities\Nodes, press Add.
32. Under ⑥ select node 3 (*8000,0,0*) with a (LC), then (RC).
33. Press OK.

Defining Loadcase 1—Fixed force on node 2

34. Under Loadcases: **Loadcases** select New → Static.
35. Set Name = fixed_force_ode_2.
36. Press Loads. Untick force_node_3. Press OK.
37. Set Stepping Procedure\#Steps = 1. Press OK.

Defining Loadcase 2—Fixed force on node 3

38. Under Loadcases: **Loadcases** select New → Static.
39. Set Name = fixed_force_node_3.
40. Press Loads. Untick force_node_2. Press OK.
41. Set Stepping Procedure\#Steps = 1 Press OK.

Running the two jobs

Job 1

42. Under Jobs: **Job** select Structural. Set Name = force_node_2.
43. Under Available, select fixed_force_node_2.
44. Press Initial Loads. Untick force_node_3. Press OK.

45. Press Check; See in ⑧ if there are any errors.
46. If there are none, press Run.
    47. In 'Run Job', press Advanced Job Submission.
        48. Press Save Model.
        49. Press Write Input File. Press OK.
        50. Press Submit 1.
    51. Wait until Status = Complete. Press OK.
52. Press OK.

Job 2

53. Under Jobs: **Job** select Structural. Set Name = force_node_3.
    54. Under Available, select fixed_force_node_3.
    55. Press Initial Loads. Untick force_node_2. Press OK.
    56. Press Check; See in ⑧ if there are any errors.
    57. If there are none, press Run.
        58. In 'Run Job', press Advanced Job Submission.
            59. Press Save Model.
            60. Press Write Input File. Press OK.
            61. Press Submit 1.
        62. Wait until Status = Complete.
        63. Press Open Post File (Model Plot Results Menu).

Viewing the model

64. Under Deformed Shape\Style, select Deformed and Original.
    65. Under Scalar Plot\Style, select Numerics.
        66. Press Scalar and select Displacement Y (or Displacement X). Press OK.
    67. Press OK.

Open Job truss_bridge_force_node_2

68. Under File → Results → Open. Open truss_bridge_force_node_2.t16
    69. In the window 'Model Plot Results', under Scalar Plot\Scalar, select Displacement Y (or Displacement X).
    70. Press OK.

**Results**

The vertical deformation of the structure is obtained for the two load cases as:

(I) $u_Y = -0.48$
(II) $u_Y = -0.97$

### 2.3.2 *Transmission Tower Structure*

**Problem Description**

Given is a simplified, i.e. plane, transmission tower structure. The transmission tower structure is idealized in the $X$–$Y$ plane based on 177 rod elements which are connected by 87 nodes. Consider the following numerical values for the material and geometrical parameters: $E = 200000$ and $A = 10$. All rod elements are connected as shown in Fig. 2.20.

The load resulting from the transmission cables is approximated by vertical forces of $F_Y = -100$. Determine the deformation at the load application points.

**Marc Solution**

Save as 'truss_tower'

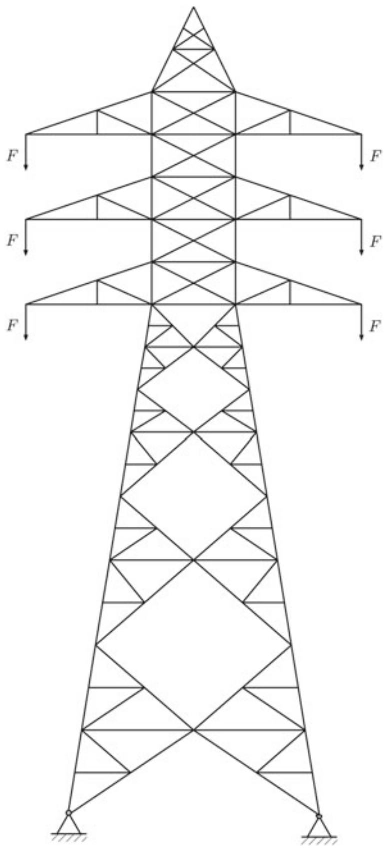



**Fig. 2.20** Idealized transmission tower structure

Constructing the mesh

1. Under Geometry&Mesh : **Basic Manipulation** select Geometry and Mesh.

2. Under Mesh\Nodes, select Add. Add the following nodes:

| Node Nr. | Node coordinates | Node Nr. | Node coordinates |
|---|---|---|---|
| 1 | *(6.000, 0.000, 0)* | 45 | *(5.330, 4.000, 0)* |
| 2 | *(2.000, 12.00, 0)* | 46 | *(4.750, 7.500, 0)* |
| 3 | *(-1.00, 12.00, 0)* | 47 | *(4.330, 10.00, 0)* |
| 4 | *(4.000, 12.00, 0)* | 48 | *(0.330, 2.000, 0)* |
| 5 | *(7.000, 12.00, 0)* | 49 | *(1.000, 6.000, 0)* |
| 6 | *(0.700, 12.00, 0)* | 50 | *(1.500, 9.000, 0)* |
| 7 | *(5.300, 12.00, 0)* | 51 | *(2.330, 9.500, 0)* |
| 8 | *(2.000, 13.00, 0)* | 52 | *(2.125, 8.250, 0)* |
| 9 | *(4.000, 13.00, 0)* | 53 | *(2.125, 6.750, 0)* |
| 10 | *(2.000, 14.00, 0)* | 54 | *(1.830, 5.000, 0)* |
| 11 | *(4.000, 14.00, 0)* | 55 | *(1.830, 3.000, 0)* |
| 12 | *(-1.00, 14.00, 0)* | 56 | *(1.500, 1.000, 0)* |
| 13 | *(7.000, 14.00, 0)* | 57 | *(1.840, 11.00, 0)* |
| 14 | *(0.700, 14.00, 0)* | 58 | *(2.500, 11.50, 0)* |
| 15 | *(5.300, 14.00, 0)* | 59 | *(2.330, 10.50, 0)* |
| 16 | *(2.000, 15.00, 0)* | 60 | *(1.920, 11.50, 0)* |
| 17 | *(4.000, 15.00, 0)* | 61 | *(1.750, 10.50, 0)* |
| 18 | *(2.000, 16.00, 0)* | 62 | *(1.580, 9.500, 0)* |
| 19 | *(4.000, 16.00, 0)* | 63 | *(1.375, 8.250, 0)* |
| 20 | *(0.700, 16.00, 0)* | 64 | *(1.125, 6.750, 0)* |
| 21 | *(5.300, 16.00, 0)* | 65 | *(0.830, 5.000, 0)* |
| 22 | *(2.000, 17.00, 0)* | 66 | *(0.495, 3.000, 0)* |
| 23 | *(4.000, 17.00, 0)* | 67 | *(0.165, 1.000, 0)* |
| 24 | *(3.000, 19.00, 0)* | 68 | *(5.660, 2.000, 0)* |
| 25 | *(2.500, 18.00, 0)* | 69 | *(5.000, 6.000, 0)* |
| 26 | *(3.500, 18.00, 0)* | 70 | *(4.500, 9.000, 0)* |
| 27 | *(2.750, 18.50, 0)* | 71 | *(4.166, 11.00, 0)* |
| 28 | *(3.250, 18.50, 0)* | 72 | *(4.165, 3.000, 0)* |
| 29 | *(-1.00, 16.00, 0)* | 73 | *(4.500, 1.000, 0)* |
| 30 | *(7.000, 16.00, 0)* | 74 | *(4.165, 5.000, 0)* |
| 31 | *(0.700, 12.57, 0)* | 75 | *(3.875, 6.750, 0)* |
| 32 | *(5.300, 12.57, 0)* | 76 | *(3.875, 8.250, 0)* |
| 33 | *(0.700, 14.57, 0)* | 77 | *(3.665, 9.500, 0)* |
| 34 | *(5.300, 14.57, 0)* | 78 | *(3.665, 10.50, 0)* |
| 35 | *(0.700, 16.57, 0)* | 79 | *(3.500, 11.50, 0)* |
| 36 | *(5.300, 16.57, 0)* | 80 | *(4.080, 11.50, 0)* |
| 37 | *(0.000, 0.000, 0)* | 81 | *(4.248, 10.50, 0)* |
| 38 | *(3.000, 11.00, 0)* | 82 | *(4.415, 9.500, 0)* |
| 39 | *(3.000, 9.000, 0)* | 83 | *(4.625, 8.250, 0)* |
| 40 | *(3.000, 2.000, 0)* | 84 | *(4.875, 6.750, 0)* |
| 41 | *(3.000, 6.000, 0)* | 85 | *(5.165, 5.000, 0)* |
| 42 | *(0.660, 4.000, 0)* | 86 | *(5.495, 3.000, 0)* |
| 43 | *(1.250, 7.500, 0)* | 87 | *(5.830, 1.000, 0)* |
| 44 | *(1.660, 10.00, 0)* | | |

3. Under Mesh\Elements, select Line (2).
4. Press Add.
5. In ⑥ connect the nodes as shown in Fig. 2.21.
6. Press OK.

Alternatively, one can also generate the model by using a procedure file (in this case ’trans_tower.proc’), a text file containing the node coordinates and the various element configurations, see Sect. 9.1 for further details.

Setting the Geometric Properties

7. Under Geometric Properties : **Geometric Properties** select New (Structural).
8. Select 3D → Truss (see Fig. 2.4).
9 Set Properties\Area = 10.
10. Under Entities\Elements, press Add.
11. In ⑦, press the first symbol ‘All Existing’.
12. Press OK.

Setting the Material Properties

13. Under Material Properties : **Material Properties** select New → Finite Stiffness → Standard.
14. Set Other Properties\Young’s Modulus = 200000 (see Fig. 2.6).
15. Under Entities\Elements, press Add.
16. In ⑦, press the first symbol ‘All Existing’.
17. Press OK.

Setting the Boundary Conditions

Boundary Supports

18. Under Boundary Conditions : **Boundary Conditions** select New (Structural) → Fixed Displacement (see Fig. 2.7).
19. Under Properties tick Displacement X and Displacement Y.
20. Under Entities\Nodes, press Add.
21. Under ⑥ select the bottom left and right node (*0,0,0* and *6,0,0*) with a (LC), then (RC).
22. Press OK.

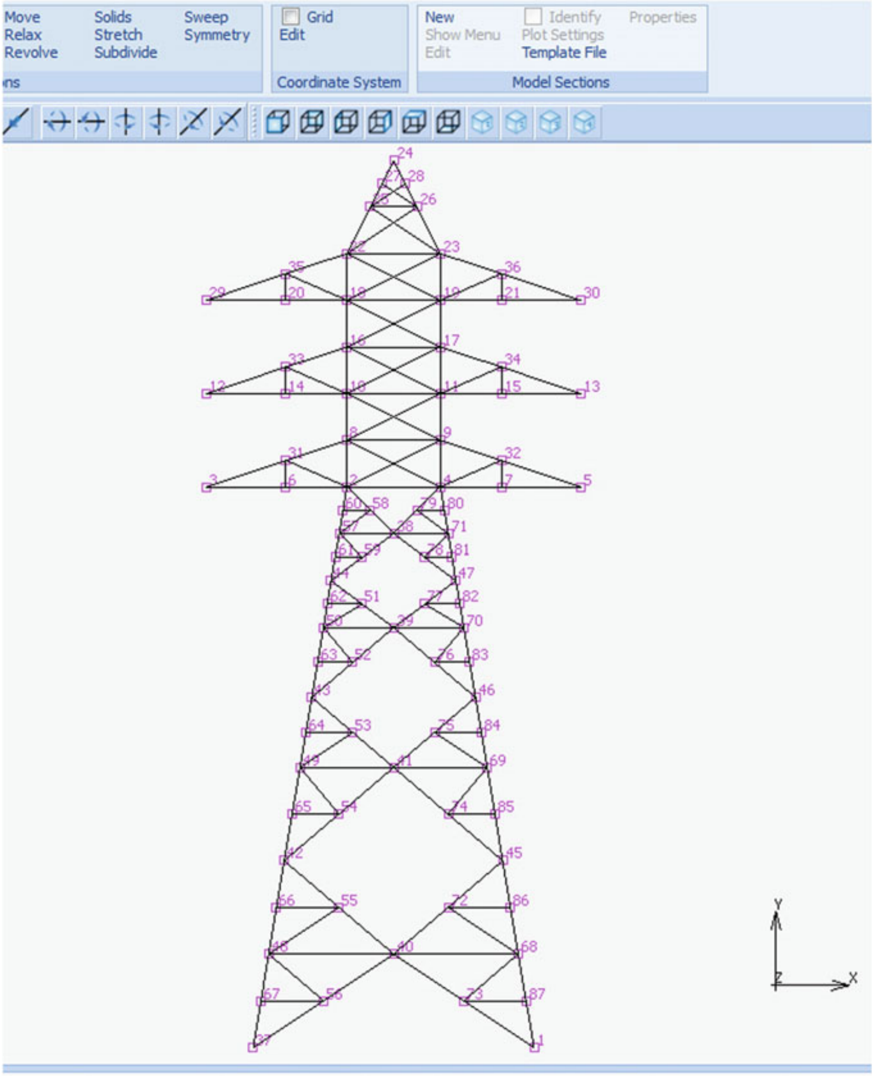



**Fig. 2.21** Transmission tower mesh

Point Loads

23. Under Boundary Conditions: **Boundary Conditions** select New (Structural) → Point Load.

24. Under Properties, tick Force Y. Set at −100.

25. Under Entities\Nodes, press Add.

26. Under ⑥ select nodes 3, 5, 12, 13, 29 and 30 with a (LC), then (RC).
27. Press OK.

Running the Job

28. Under Jobs: **Job** → Structural.
29. Press Check; See in ⑧ if there are any errors.
30. If there are none, press Run.
31. In 'Run Job', press Advanced Job Submission (see Fig. 2.9).
32. Press Save Model.
33. Press Write Input File Press OK.
34. Press Submit 1.
35. Wait until Status = Complete.
36. Press Open Post File (Model Plot Results Menu).

Viewing the model

37. Under Deformed Shape\Style, select Deformed and Original.
38. Under Scalar Plot\Style, select Numerics (see Fig. 2.10).
39. Press Scalar and select Displacement Y.
40. Press OK.

**Results**

The deformation on the load application points is found to be $u_Y = -0.006$.

**Additional Questions**

1. Determine the reaction forces at the ground supports.

## 2.4 Supplementary Problems

### *2.4.1 Truss Structure with Six Members*

**Problem Description**

Given is a plane truss structure as shown in Fig. 2.22. The members have a uniform cross-sectional area $A = 5$ and YOUNG's modulus $E = 30$. The length of each member can be taken from the figure ($L = 2$). The structure is fixed at its left-hand side and loaded by

- two prescribed displacements $u_0 = 0.2$ and $2u_0$ at the very right-hand corner,
- a vertical point load $F_0 = 30$.

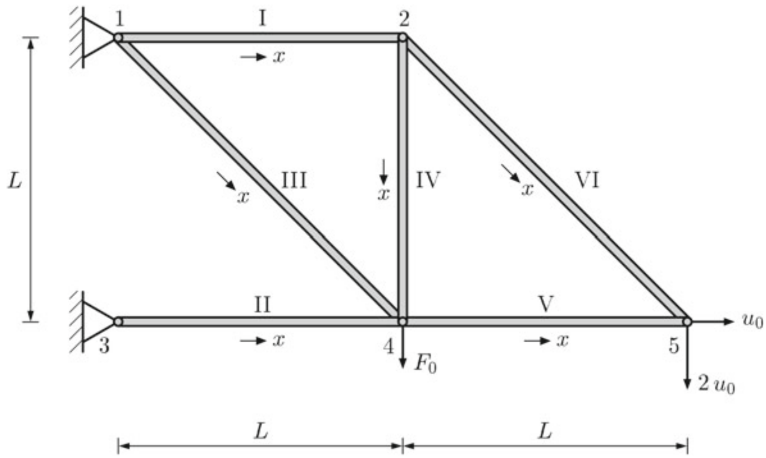



**Fig. 2.22** Truss structure composed of six axial members

Model the truss structure with six linear finite elements (MSC Marc element type 9) and determine

- the displacements of the nodes,
- the reaction forces at the supports and nodes where displacements are prescribed,
- the strain, stress, and normal force in each element.
- Check the obtained results based on the global force equilibrium.

Assume that all the numbers are given in consistent units. Simplify all your results for the following special cases:

(a) $u_0 = 0$,
(b) $F_0 = 0$.

http://www.springer.com/978-3-319-71914-6

A First Introduction to the Finite Element Analysis
Program MSC Marc/Mentat
Öchsner, A.; Öchsner, M.
2018, XV, 158 p. 88 illus., 43 illus. in color., Hardcover
ISBN: 978-3-319-71914-6