4. TRUSS EXAMPLE: Analysis of an overhead hoist

The example of an overhead hoist, shown in Figure 4-1, is used to illustrate the basics of ABAQUS/CAE modeling process by using the Model Tree and showing the basic steps used to create and analyze a simple model. The hoist is modeled as a simple, pin-jointed truss that is constrained at the left-hand end and mounted on rollers at the right-hand end. The members can rotate freely at the joints. The truss is prevented from moving out of plane. A simulation is performed to determine the structure's deflection and the peak stress in its members when a 10 kN load is applied as shown in Figure 4-1.

All members are circular steel rods, 5 mm in diameter.

Material properties
General properties:
$\rho = 7800\ kg/m^3$
Elastic properties:
$E = 200 \times 10^9\ Pa$
$\nu = 0.3$

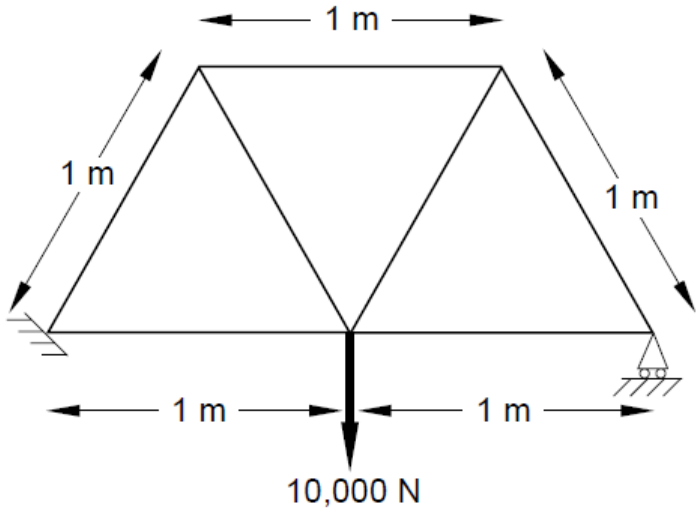


Figure 4–1 Schematic of an overhead hoist.

For the overhead hoist example, you will perform the following tasks:

- Sketch the two-dimensional geometry and create a part representing the frame.
- Define the material properties and section properties of the frame.
- Assemble the model.
- Configure the analysis procedure and output requests.
- Apply loads and boundary conditions to the frame.
- Mesh the frame.
- Create a job and submit it for analysis.
- View the results of the analysis.

NOTE:
Abaqus has NO built-in system of units. Do NOT include unit names or labels when entering data in Abaqus. All input data must be specified in consistent units. The SI system of units is used throughout this guide.

You also need to decide which coordinate system to use. The global coordinate system in Abaqus is a right-handed, rectangular (Cartesian) system. For this example define the global 1-axis to be the horizontal axis of the hoist and the global 2-axis to be the vertical axis. The global 3-axis is

The title block at the bottom of the viewport indicates the following:

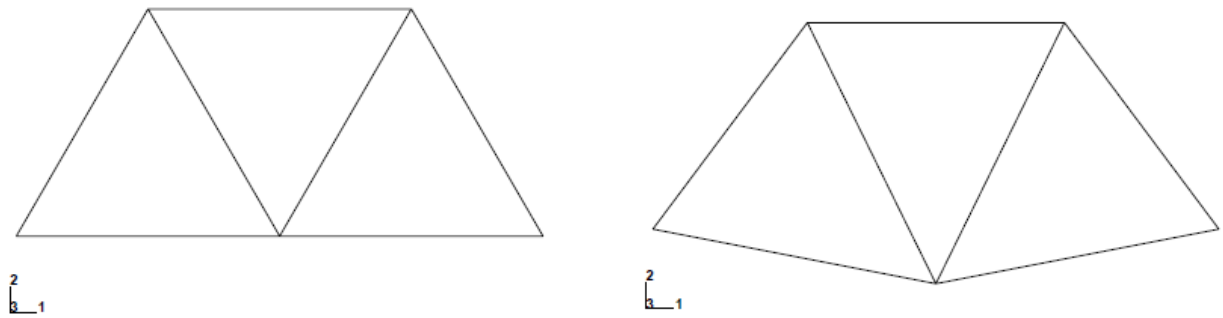
- The description of the model (from the first line of the *HEADING option in the input file).
- The name of the output database (from the name of the analysis job).
- The product name (Abaqus/Standard or Abaqus/Explicit) and version used to generate the output database.
- The date the output database was last modified.

The state block at the bottom of the viewport indicates the following:

- Which step is being displayed.
- The increment within the step.
- The step time.

The view orientation triad indicates the orientation of the model in the global coordinate system.

You will now display the undeformed model shape and use the plot options to enable the display of node and element numbering in the plot. From the main menu bar, select **Plot→Undeformed Shape**; or use the  tool in the toolbox. Abaqus/Viewer displays the undeformed model shape, as shown in Figure 4–6a.



(a) (b)

Figure 4–6 (a) Undeformed model shape; and (b) Deformed model shape.

To display node numbers

1. From the main menu bar, select **Options→Common.** The **Common Plot Options** dialog box appears.
2. Click the **Labels** tab.
3. Toggle on **Show node labels**.
4. Click **Apply**.

Abaqus/Viewer applies the change and keeps the dialog box open.

To display element numbers

SAMPLE REPORT FILE

Field Output reported at nodes for part: FRAME

Node Label	U.Magnitude @Loc 1	U.U1 @Loc 1	U.U2 @Loc 1
1	254.712E-06	-588.233E-15	-254.712E-06
2	294.116E-06	147.058E-06	-254.712E-06
3	472.726E-06	73.5291E-06	-466.972E-06
4	147.058E-06	147.058E-06	-5.E-33
5	0.	-0.	-5.E-33
Minimum	0.	-588.233E-15	-466.972E-06
At Node	5	1	3
Maximum	472.726E-06	147.058E-06	-5.E-33
At Node	3	4	5

Field Output reported at nodes for part: FRAME-1

Node Label	RF.Magnitude @Loc 1	RF.RF1 @Loc 1	RF.RF2 @Loc 1
1	0.	0.	0.
2	0.	0.	0.
3	0.	0.	0.
4	5.E+03	0.	5.E+03
5	5.E+03	2.27374E-12	5.E+03
Minimum	0.	0.	0.
At Node	3	4	3
Maximum	5.E+03	2.27374E-12	5.E+03
At Node	5	5	5
Total	10.E+03	2.27374E-12	10.E+03