

Lab 2

Creep in a Wire

Introduction

This lab is an example of high-temperature creep analysis. The wire being analyzed is 305 mm long and has a diameter of 3.175 mm. A load of 50 kg is applied to the end of the wire for a period of 180,000 seconds (50 hours). The wire is made of an isotropic elastic material, with a Young's modulus of 138 GPa and a Poisson's ratio of 0.3. The Mises creep potential and uniaxial creep behavior is defined by $\dot{\epsilon}^{cr} = A\sigma^n$, where A is 1.7828×10^{-17} per sec (stress in MPa) and $n = 5$. These values are typical of structural steel at a fairly high temperature.

Recall that Abaqus/CAE does not use specific units, but the units must be consistent throughout the model. Note that SI units are used in this lab: N, mm, tonne, and sec. With this choice of units, stresses are expressed in MPa.

Preliminaries

1. In the **Start Session** dialog box, underneath **Create Model Database**, click **With Standard/Explicit Model**.
2. To create a model, select **Model**→**Create** from the main menu bar and enter the name **WireCreep** in the **Edit Model Attributes** dialog box. Click **OK** (you could also rename Model-1 to **WireCreep**).
3. To save the model database, select **File**→**Save As** from the main menu bar and type the file name **WireCreep** in the **Save Model Database As** dialog box. Click **OK**.

The **.cae** extension is added to the file name automatically.

Creating a part

In this section, you will create a three-dimensional, deformable wire by sketching a one-dimensional profile of the wire.

1. Abaqus/CAE automatically loads the Part module. Any other module can be accessed from the **Module** list located in the context bar, as shown in Figure L2–1.

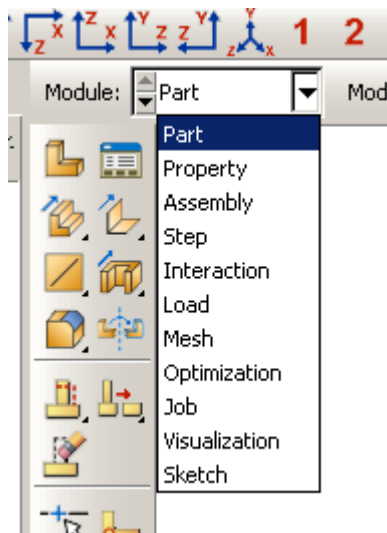


Figure L2–1. Module list

2. From the main menu bar, select **Part**→**Create** to create a new part. In the **Create Part** dialog box that appears, name the part **Wire**, and specify an approximate size of **500**. Accept the settings of a three-dimensional, deformable body with a wire, Planar base feature. Click **Continue**.
Abaqus/CAE displays text in the prompt area near the bottom of the window to guide you through the procedure, as shown in Figure L2–2. Click the cancel button to cancel the current task; click the backup button to cancel the current step in the task and return to the previous step.

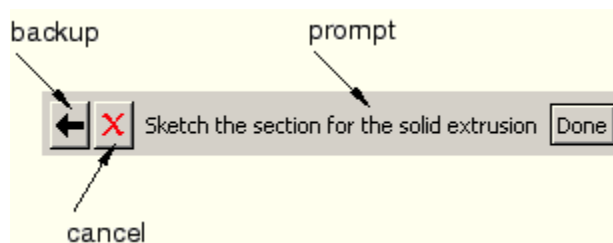


Figure L2–2. Prompt area.

- The Sketcher toolbox appears in the left side of the main window, and the Sketcher grid appears in the viewport.
3. To sketch the profile of the wire, you need to select the Create Lines: Connected sketch tool, as shown in Figure L2–3.

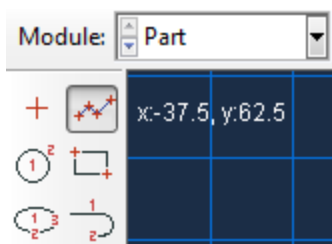



Figure L2–3. Create Lines: Connected sketch tool

4. In the viewport, sketch the wire as follows:
 - a. When Abaqus prompts for picking the starting point for the line, enter (0,-200). Then enter the end point for the line as (0,105).
 - b. Click mouse button 2 anywhere in the viewport to exit the Create Lines: Connected sketch tool.
 - c. Use the dimension tool  to dimension the wire. The wire should have a dimension of **305** mm. When dimensioning the line, simply select the line, click mouse button 1 to position the dimension text, and then enter the new dimension in the prompt area. The final sketch is shown in Figure L2–4.

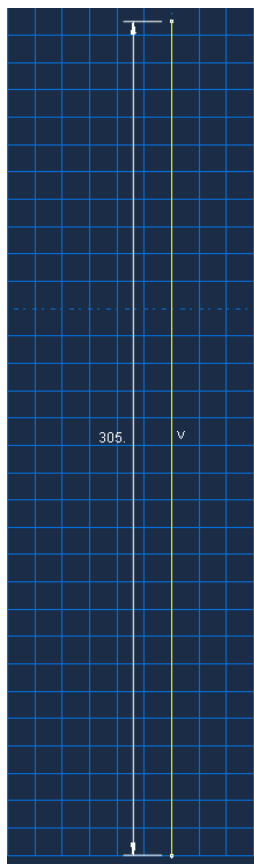


Figure L2–4. Sketch of the wire.

5. Click **Done** in the prompt area to exit the sketcher.

Abaqus/CAE displays the new part, as shown in Figure L2–5.

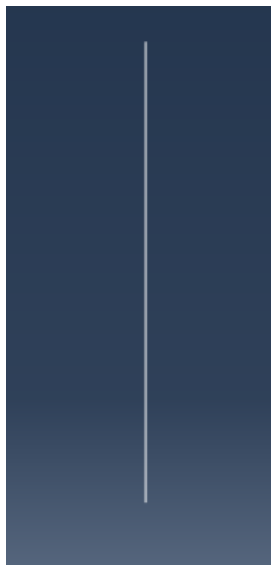


Figure L2–5. Wire part.

Creating a material definition

You will now create an isotropic elastic material, with a Young's modulus of 138 GPa and a Poisson's ratio of 0.3.

To define a material:

1. In the **Module** list located in the context bar, select **Property** to enter the Property module.
2. From the main menu bar, select **Material**→**Create** to create a new material.
3. In the **Edit Material** dialog box that appears, name the material **Steel1**. Notice the various options available in this dialog box.

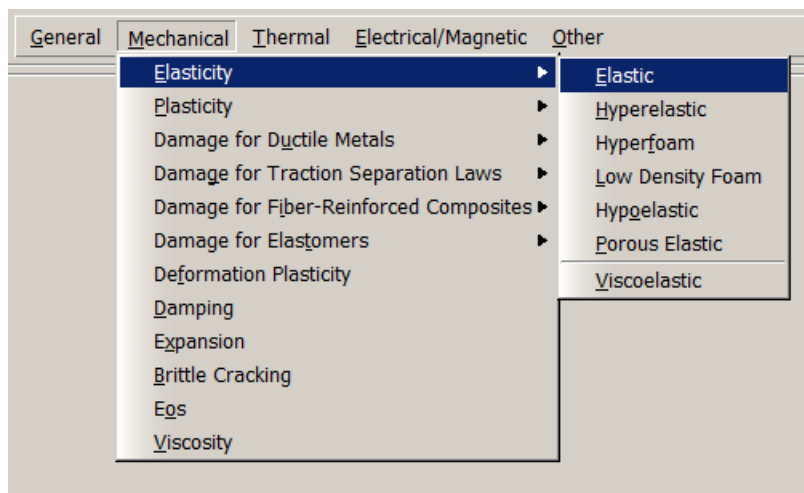


Figure L2–6. Material pull-down menu.

4. From the material editor's menu bar, select **Mechanical**→**Elasticity**→**Elastic**, as shown in Figure L2–6.

Abaqus/CAE displays the **Elastic** data form.

5. Enter a value of **138E3** for Young's modulus and a value of **0.3** for Poisson's ratio in the respective fields, as shown in Figure L2–7. Use **[Tab]** to move between cells, or use the mouse to select a cell for data entry.

Data		
	Young's Modulus	Poisson's Ratio
1	138E3	0.3

Figure L2–7. Material editor.

6. From the material editor's menu bar, select **Mechanical**→**Plasticity**→**Creep**, Abaqus/CAE displays the **Creep** data form.
7. Enter a value of **1.7828E-17** for the Power Law Multiplier, **5** for the Eq Stress Order and **0** for the Time Order in the respective fields, as shown in Figure L2-8.
8. Click **OK** to exit the material editor.

Data			
	Power Law Multiplier	Eq Stress Order	Time Order
1	1.7828E-17	5	0

Figure L2–8. Material editor.

Creating the wire profile

The circular section profile for the wire will be defined now.

To create the profile

- a. From the main menu bar, select **Profile**→**Create**.
- b. In the **Create Profile** dialog box that appears:
 - a. Name the profile **CircularSection**.
 - b. Select the **Circular** shape
 - c. Click **Continue**.
- c. In the **Edit Profile** dialog box that appears:
 - a. Enter values of 1.5875 mm for **r**
 - b. Click **OK**.

Defining and assigning section properties

Next, you will create a beam section and assign it to the wire. The section will refer to the material **Steel** and the **CircularSection** profile that you just created.

To define the beam section:

1. From the main menu bar, select **Section→Create**.
2. In the **Create Section** dialog box that appears:
 - a. Name the section **WireSection**.
 - b. Select the Category and Type **Beam**.
 - c. Click **Continue**.
3. In the **Edit Beam Section** dialog box that appears:
 - a. Accept the selection of **CircularSection** for the **Profile name** and **Steel** for the **Material** associated with the section.
 - b. Click **OK**.

To assign the section definition to the bar:

1. From the main menu bar, select **Assign→Section**.
Abaqus/CAE displays text in the prompt area to guide you through the procedure.
2. Click anywhere on the wire to select it as the region to which the section will be assigned.
3. Click mouse button 2 in the viewport or click **Done** in the prompt area to accept the selected geometry.
The section assignment editor appears.
4. In the **Edit Section Assignment** dialog box, accept the default selection of **WireSection** as the section definition, and click **OK**.
Abaqus/CAE colors the part green to indicate that the section has been assigned.

Assigning beam section orientation

The orientation of section of the wire with respect to the global axes will be assigned now.

To assign the beam section orientation

1. From the main menu bar, select **Assign→Beam Section Orientation**.
2. Click anywhere on the wire to select it as the region to which the orientation will be assigned.
3. Click **Done** in the prompt area.
4. Accept the default n1 direction. Press Enter and then click **OK** in the prompt area, and then click **Done**.

Assembling the model

The assembly for this analysis consists of a single instance of the part **Wire**.

To assemble the model:

1. In the **Module** list located in the context bar, select **Assembly** to enter the Assembly module.
2. From the main menu bar, select **Instance→Create**.
3. In the **Create Instance** dialog box, select **Wire** and click **OK**.
Abaqus/CAE displays the new part instance in the viewport.

Configuring the analysis

A quasi-static stress analysis in Abaqus/Standard is used to analyze problems with time-dependent material response like in creep. In this analysis, we will first apply the 50 kg load in a general, static analysis step. We will then create another general, visco step with a time period of 180,000 sec during which creep due to the applied load takes place.

Abaqus/CAE generates the initial step automatically, but you must create the analysis steps yourself.

To create a general, static analysis step:

1. In the **Module** list located in the context bar, select **Step** to enter the Step module.
2. From the main menu bar, select **Step**→**Create** to create a step.
3. In the **Create Step** dialog box that appears:
 - a. Name the step **AppliedLoad**.
 - b. From the list of available general procedures in the **Create Step** dialog box, select **Static, General** if it is not already selected.
 - c. Click **Continue**.
4. In the **Edit Step** dialog box that appears:
 - a. In the **Description** field of the **Basic** tabbed page, enter **Load the bottom end of the wire**.
 - b. Enter a Time period value of **1** sec.
 - c. Click **OK** to create the step and to exit the step editor.

To create a general, visco analysis step:

1. From the main menu bar, select **Step**→**Create** to create a step.
2. In the **Create Step** dialog box that appears:
 - a. Name the step **Creep**.
 - b. From the list of available general procedures in the **Create Step** dialog box, select **Visco**.
 - c. Click **Continue**.
3. In the **Edit Step** dialog box that appears:
 - a. In the **Description** field of the **Basic** tabbed page, enter **Occurrence of creep**.
 - b. Enter a **Time period** value of **180,000** sec.
 - c. Under the **Incrementation** tabbed page, enter a **Creep/swelling/viscoelastic strain error tolerance** of **5E-006**.
 - d. Click **OK** to create the step and to exit the step editor.

Applying boundary condition to the model

Next, you will define the boundary condition, and load that will be applied during the general, static step. The effect of this load will be propagated to general, visco step that follows during which creep occurs.

1. In the **Module** list located in the context bar, select **Load** to enter the Load module.
2. From the main menu bar, select **BC→Create**.
3. In the **Create Boundary Condition** dialog box that appears:
 - a. Name the boundary condition **FixedTop**.
 - b. Select **Initial** as the step in which the boundary condition will be activated.
 - c. In the **Category** list, accept the default category selection **Mechanical**.
 - d. In the **Types for Selected Step** list, select **Symmetry /Antisymmetry/ Encastre**.
 - e. Click **Continue**.
4. Click on the top node of the wire.
5. Click mouse button 2 in the viewport or click **Done** in the prompt area to accept the selected geometry.

The **Edit Boundary Condition** dialog box appears. When you are defining a boundary condition in the initial step, all six degrees of freedom are unconstrained by default.

6. In the **Edit Boundary Condition** dialog box:
 - a. Toggle on **ENCASTRE (U1 = U2 = U3 = UR1 = UR2 = UR3 = 0)**
 - b. Click **OK** to create the boundary condition definition and to exit the editor.

Applying load to the model

Now, you will apply the 50 kg ($50 \text{ kg} \times 9.8 \text{ m/s}^2 = 490 \text{ N}$) load at the bottom of the wire.

1. From the main menu bar, select **Load→Create**.
2. In the **Create Load** dialog box:
 - a. Name the load **PointLoad**.
 - b. Select **AppliedLoad** as the **Step** in which the load will be applied.
 - c. Select **Mechanical** as the **Category** as and **Concentrated force** as the **Types for Selected Step**.
 - d. Click **Continue**.
3. Click on the bottom node of the wire.
4. Click mouse button 2 in the viewport or click **Done** in the prompt area to accept the selected geometry.
5. In the **Edit Load** dialog box, enter a value of **-490 N** for CF2, and **0 N** for CF1 and CF3.
6. Accept the default selection of **(Ramp)** for the **Amplitude**. This will cause the load to be applied as a ramp during the AppliedLoad general, static step over a time period of 1 sec.
7. Click **OK**.

Meshing the model

You use the Mesh module to generate the finite element mesh. You can choose the meshing technique that Abaqus/CAE will use to create the mesh, the element shape, and the element type. Abaqus/CAE uses a number of different meshing techniques. The default meshing technique assigned to the model is indicated by the color of the model when you enter the Mesh module; if Abaqus/CAE displays the model in orange, it cannot be meshed without assistance from the user.

To assign an Abaqus element type:


1. In the **Module** list located in the context bar, select **Mesh** to enter the Mesh module.
2. In the context bar, select **Part** as the displayed object.
3. From the main menu bar, select **Mesh→Element Type**.
4. Click anywhere on the wire to select it as the region to be assigned the element type.
5. Click on **Done**.
6. In the **Element Type** dialog box, accept the following default selections that control the elements that are available for selection:
 - **Standard** is the default **Element Library** selection.
 - **Linear** is the default **Geometric Order**.
 - **Beam** is the default **Family** of elements.

A description of the element type B31 appears at the bottom of the dialog box. Abaqus/CAE will now mesh the part with B31 elements.

7. Click **OK** to assign the element type and to close the dialog box.
8. Click **Done** in the prompt area.

To mesh the model:

1. From the main menu bar, select **Seed→Part** to seed the part.
The **Global Seeds** dialog box appears. The default global element size is based on the size of the part.
2. In the **Global Seeds** dialog box, enter an approximate global size of **10** and click **OK**.

Tip: You can toggle on persistent display of seeds by clicking  in the **Visible Objects** toolbar.

3. From the main menu bar, select **Mesh→Part** to mesh the part.
4. Click **Yes** in the prompt area or click mouse button 2 in the viewport to confirm that you want to mesh the part instance.

Abaqus/CAE meshes the part instance and displays the resulting mesh.

Creating and submitting an analysis job

The definition of the model **WireCreep** is now complete. Next, you will create and submit an analysis job to analyze the model.

To create and submit an analysis job:

1. In the **Module** list located in the context bar, select **Job** to enter the Job module.
2. From the main menu bar, select **Job**→**Manager**.
3. From the buttons on the bottom of the **Job Manager**, click **Create** to create a job.
4. In the **Create Job** dialog box that appears, name the job **WireCreep** and select the model **WireCreep**. Click **Continue**.

The job editor appears.

5. In the **Description** field of the **Edit Job** dialog box, enter **Lab 2**.
6. Click the tabs to see the contents of each folder of the job editor and to review the default settings. Click **OK** to accept the default job settings.
7. From the buttons on the right side of the **Job Manager**, click **Submit** to submit your job for analysis. The status field will show **Running**.

When the job completes successfully, the status field will change to **Completed**. You are now ready to view the results of the analysis in the Visualization module.

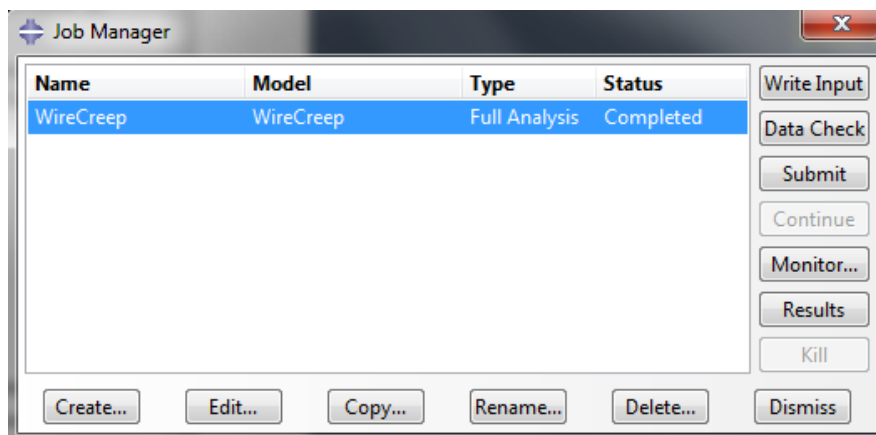





Figure L2–9. Job status in the Job Manager.

Viewing the analysis results

You are now ready to view the results of the analysis in the Visualization module.

1. Click **Results** in the **Job Manager** to enter the Visualization module.
Abaqus/CAE switches to the Visualization module, opens the output database created by the job (**WireCreep.odb**), and displays the undeformed shape of the model.
2. From the main menu bar select **View**→**ODB Display Options**, and toggle on **Render beam profiles** to view the circular wire profile in 3D. Click **OK**.
3. In the toolbox, click  to view a deformed model shape plot.
4. In the toolbox, click  (or select **Plot**→**Contours**→**On Deformed Shape** from the main menu bar) to view a contour plot of the von Mises stress, as shown in Figure L2–10. You can also click on  to view the time history animation.

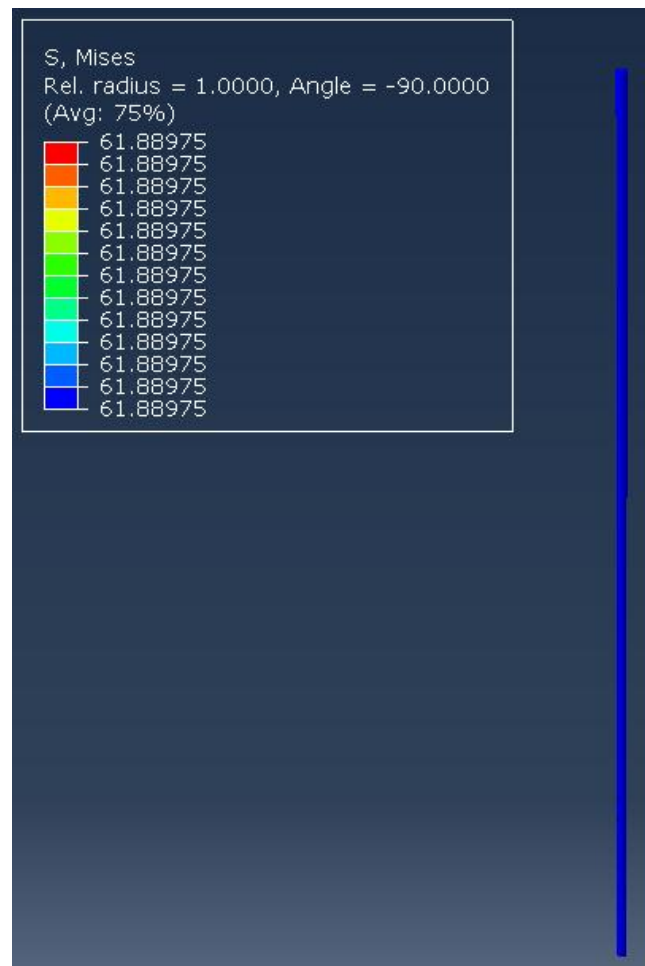


Figure L2–10. Mises stress contour plot.

- From the main menu bar, select **Tools**→**XY Data** → **Create**. Select **odb field output** and click continue. In the **XY Data from ODB Field Output** dialog box, under the **Variables** tab, change the position to **Unique nodal**, expand **E: Strain components** and check **E11**.
- Click on to the **Element/Nodes** tab and select the **Pick from viewport** selection method. Click on **Edit Selection** and select the bottom node of the wire from the viewport. Click **Done** in the prompt area.
 - Click **Plot** in the **XY Data from ODB Field Output** dialog box. Click **Dismiss**. Strain vs. Time plots (log and linear time scale) for the bottom node of the wire are displayed as shown in Figure L2–11 (a) and (b).
As shown in the figure, the wire elongates slightly when the load is applied to it in the first analysis step. After that, the wire experiences creep deformation i.e. it undergoes a slow, continuous deformation over a period of 50 hours.
 - Perform the same creep analysis by doubling and halving the load. The results will be discussed in the quiz.

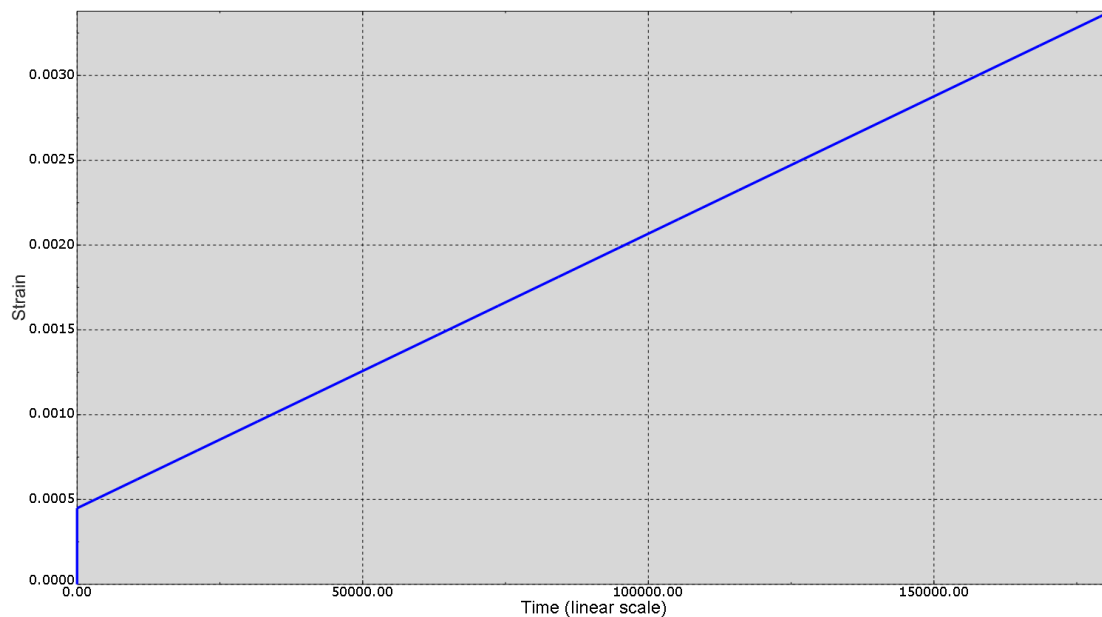


Figure L2-11 (a). Strain vs. Time plot for 490 N load

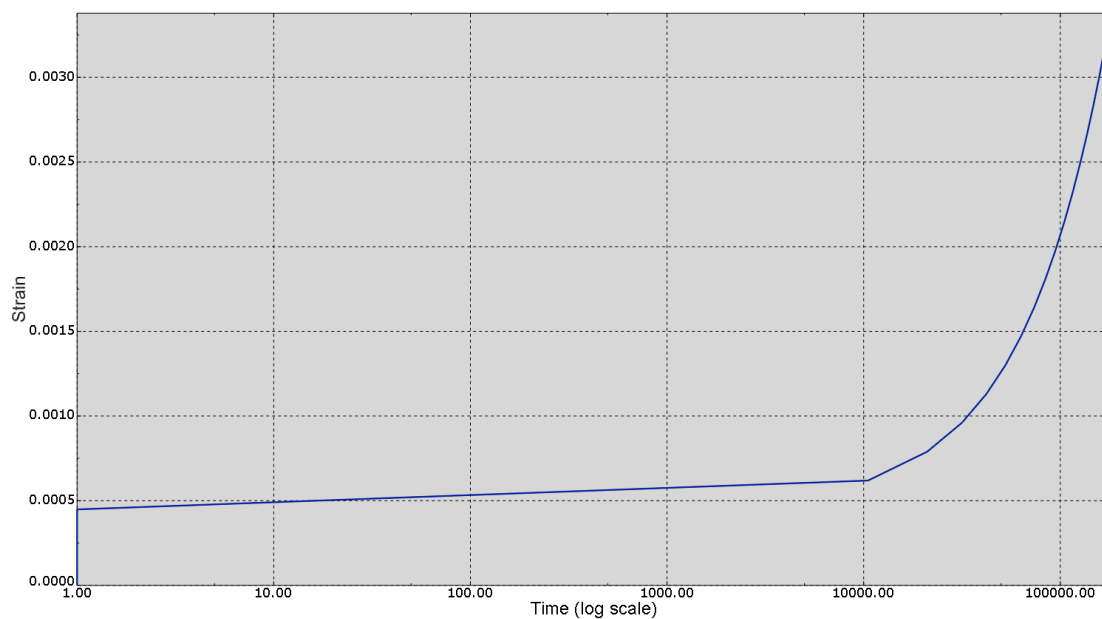



Figure L2-11 (b). Strain vs. Time plot for 490 N load

8. In the toolbar, click  (or select **File**→**Save** from the main menu bar) to save your model in a model database file.