Lab 1

Tensile Loading of a Rectangular Specimen

Introduction

During tensile loading, a specimen is subjected to tension until it fractures. During the application of tension, the elongation of the specimen is recorded against the applied force. In this lab, you will be simulating the tensile loading conditions on a rectangular bar of 10 X 10 X 100 mm dimensions. We will be applying displacement based loading and studying the behavior of the material within the elastic limit.

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 **`TENSILEBAR`** in the **Edit Model Attributes** dialog box. Click **OK**. (you could also rename Model-1 to **`TENSILEBAR`**)
3. To save the model database, select **File→Save As** from the main menu bar and type the file name **`TENSILEBAR`** 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 solid body by sketching the two-dimensional profile of the bar (a rectangle) and extruding it.

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 L1a–1

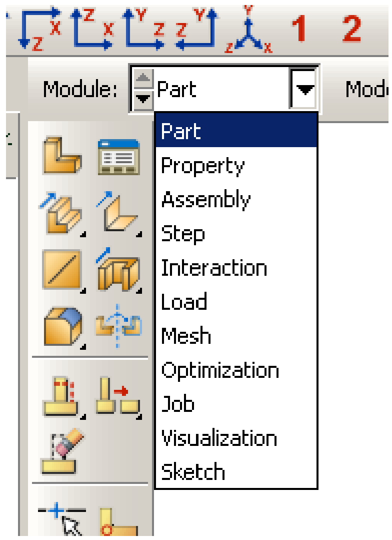


Figure L1a–1. Module list

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 **`Bar`**, and specify an approximate size of **`500`**. Accept the default settings of a three-dimensional, deformable body with a solid, extruded 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 L1a–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.

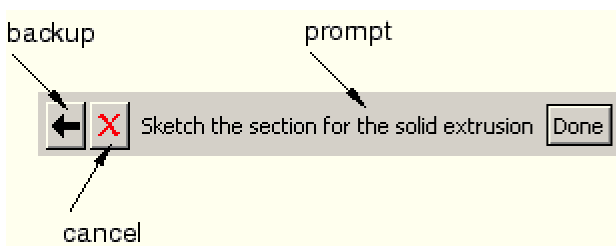


Figure L1a–2. Prompt area.

   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 bar, you need to select the rectangle drawing tool, as shown in Figure L1a–3.

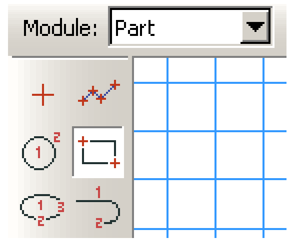


Figure L1a–3. Rectangle sketch tool.

4. In the viewport, sketch the rectangle as follows:
    a. When Abaqus prompts for picking the start corner for the rectangle, enter (0,-20). Then enter the opposite corner for the rectangle as (10,80).
    b. Click mouse button 2 anywhere in the viewport to exit the rectangle tool.
    c. The Sketcher automatically adds constraints to the sketch (in this case the four corners of the rectangle are assigned perpendicular constraints and one edge is designated as horizontal).
    d. Use the dimension tool  to dimension the top and left edges of the rectangle. The top edge should have a horizontal dimension of **10** mm, and the left edge should have a vertical dimension of **100** mm. When dimensioning each edge, 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 L1a–4.

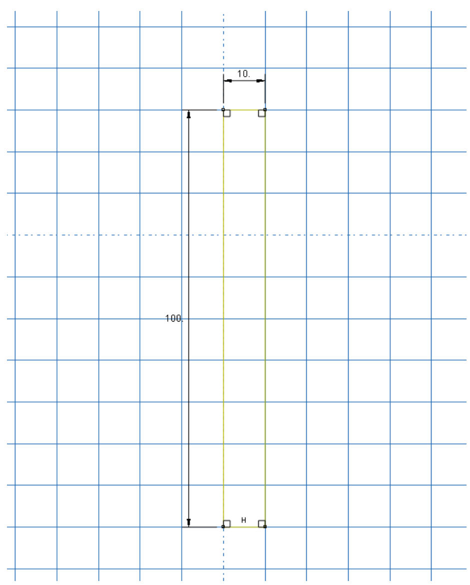


Figure L1a–4. Sketch of the rectangle.

5. Click **Done** in the prompt area to exit the sketcher.
6. Enter an extrusion depth of **10** in the **Edit Base Extrusion** dialog box, and click **OK**.

    Abaqus/CAE displays an isometric view of the new part, as shown in Figure L1a–5.

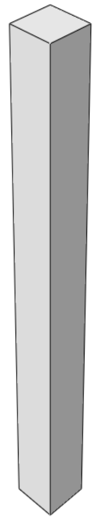


Figure L1a–5. Extruded part.

Creating a material definition

You will now create a single linear elastic material with a Young's modulus of 209×10^3 MPa and 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 `Steel`. Notice the various options available in this dialog box.

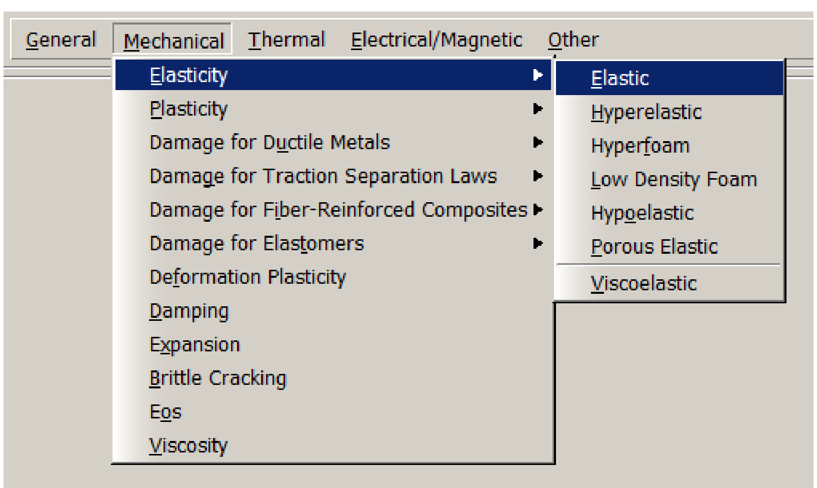


Figure L1a–6. Material pull–down menu.

4. From the material editor's menu bar, select **Mechanical→Elasticity→Elastic,** as shown in Figure L1a–6.

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

5. Enter a value of `209.E3` for Young's modulus and a value of `0.3` for Poisson's ratio in the respective fields, as shown in Figure L1a–7. Use **[Tab]** to move between cells, or use the mouse to select a cell for data entry.

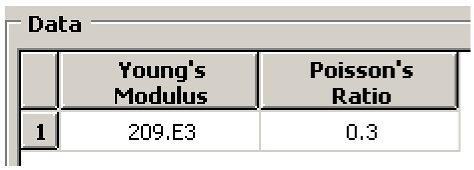


Figure L1a–7. Material editor.

6. Click **OK** to exit the material editor.

Defining and assigning section properties

Next, you will create a homogeneous solid section and assign it to the bar. The section will refer to the material `Steel` that you just created.

To define the homogeneous solid section:

1. From the main menu bar, select **Section→Create**.
2. In the **Create Section** dialog box that appears:
   a. Name the section `SolidSection`.
   b. Accept the default category **Solid** and the default type **Homogeneous**.
   c. Click **Continue**.
3. In the **Edit Section** dialog box that appears:
   a. Accept the default selection of **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 prompts in the prompt area to guide you through the procedure.

2. Click anywhere on the part 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 **SolidSection** as the section definition, and click **OK**.

   Abaqus/CAE colors the part green to indicate that the section has been assigned.

Assembling the model

The assembly for this analysis consists of a single instance of the part `Bar`.

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 `Bar` and click **OK**.

   Abaqus/CAE displays the new part instance in the viewport.
4. From the main menu bar, select **Tools→Reference Point.** Enter the coordinates of the reference point as (5, 83, 5).

Configuring the analysis

In this simulation we are interested in the static response of the bar to a tensile load applied on its top face. So, you will create a general, static analysis step, in which you will apply a displacement load to the top face of the bar.

Abaqus/CAE generates the initial step automatically, but you must create the analysis step 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 `TensileLoad`.
   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 top of the beam`.
   b. Click the **Incrementation** tab, and delete the value of `1` that appears in the **Initial** text field. Type a value of `0.1` for the initial increment size.
   c. Click the **Other** tab to see its contents; you can accept the default values provided for the step.
   d. Click **OK** to create the step and to exit the step editor.
5. Output requests are created by default once you define a step, but you can edit them if need be. From the main menu bar, select **Output→Field Output Requests→Edit→F-Output-1.** Select **Evenly spaced time intervals** in the **Frequency** drop down menu and in the **Interval** box write `20`. The output is written to the odb at every 1/20th step time increment this way.

Applying a kinematic constraint to the model

A kinematic constraint couples the motion of a collection of nodes on a surface to the rigid body motion of a reference node.

1. In the **Module** list located in the context bar, enter the **Interaction** module.
2. From the main menu bar, select **Constraint→Create** to create a constraint.
3. In the **Create Constraint** dialog box that appears:
    a. Name the constraint **`Coupling`**.
    b. From the list of types of constraints in the **Create Constraint** dialog box, select **Coupling** and hit **Continue**.
4. Select the reference point **RP-1** from the viewport to act as the constraint control point and hit **Done**.
5. Select **surface** as the constraint region type and then select the top surface of the bar. In the **Edit Constraint** dialog box that appears make sure that the coupling type selected in **kinematic** and all degrees of freedom (U1, U2, U3, UR1, UR2, and UR3) are constrained. Select **OK** to exit the edit constraint dialog box.

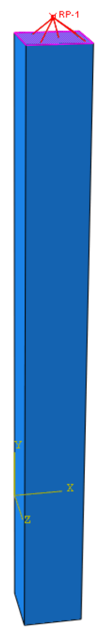


Figure L1a–8. Kinematic coupling definition.

Applying boundary conditions to the model

Next, you will define the boundary condition and loading that will be active during the **`TensileLoad`** step.

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 **`Fixed`**.
    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 **Displacement/Rotation** as the type.
    e. Click **Continue**.
4. Rotate the part so that you can pick the desired face (bottom), as shown in Figure L1a–9.

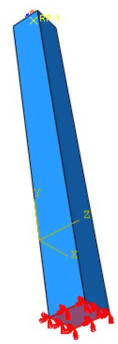


Figure L1a–9. Fixed end.

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 **U1**, **U2**, and **U3** to constrain the translational degrees of freedom (solid elements, which will be used to create the mesh, only possess translational DOFs).
    b. Click **OK** to create the boundary condition definition and to exit the editor.
7. Create another boundary condition and in the **Create Boundary Condition** dialog box that appears:
    a. Name the boundary condition **`DisplaceTop`**.
    b. Select **TensileLoad** 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 **Displacement/Rotation** as the type.
    e. Click **Continue**.
8. Select the reference point **RP-1** when prompted to select the regions for the boundary condition and click done to accept the geometry selection.

9. In the **Edit Boundary Condition** dialog box:
   a. Toggle on **U1, U2, U3, UR1**, **UR2** and **UR3** to constrain all the degrees of freedom and enter **U2 = 3** so that the top face is pulled by 3 mm.
   b. Click **OK** to create the boundary condition definition and to exit the editor.

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 the mesh controls:

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→Controls**.
4. In the **Mesh Controls** dialog box that appears, accept **Hex** as the default **Element Shape** selection.
5. Accept **Structured** as the default **Technique** selection.
6. Click **OK** to assign the mesh controls and to close the dialog box.

To assign an Abaqus element type:

1. From the main menu bar, select **Mesh→Element Type**.
2. 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**.
   - **3D Stress** is the default **Family** of elements.
3. In the lower portion of the dialog box, examine the element shape options. A brief description of the default element selection is available at the bottom of each tabbed page.
4. In the **Hex** tabbed page, select **Reduced integration**.

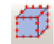
   A description of the element type C3D8R appears at the bottom of the dialog box. Abaqus/CAE will now mesh the part with C3D8R elements.
5. Click **OK** to assign the element type and to close the dialog box.

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 **`3`** 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, as shown in Figure L1a–10.

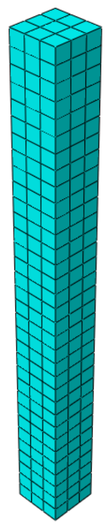


Figure L1a–10. Part instance showing the resulting mesh.

Creating and submitting an analysis job

The definition of the model **TENSILEBAR** 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 **`tensileloading`** and select the model **TENSILEBAR**. Click **Continue**.

   The job editor appears.
5. In the **Description** field of the **Edit Job** dialog box, enter **`Lab 1`**.
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.

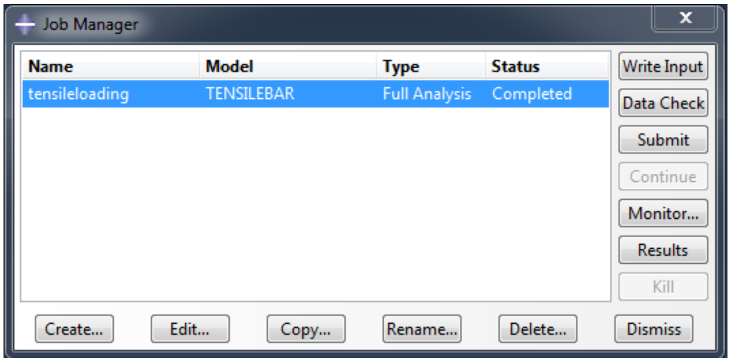


Figure L1a–11. Job status in the Job Manager.

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 (**`tensileloading.odb`**), and displays the undeformed shape of the model. In the toolbox, click  to view a deformed model shape plot. Figure L1a–12 shows a plot comparing the two shapes.

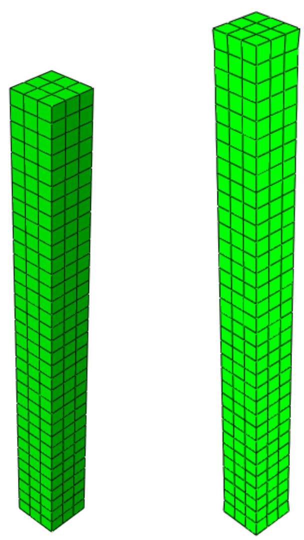


Figure L1a–12. Undeformed and deformed model shapes.

You may need to use the **Auto-Fit View** tool  to rescale the figure in the viewport.

2. 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 L1a–13. You can also click on  to view the time history animation.

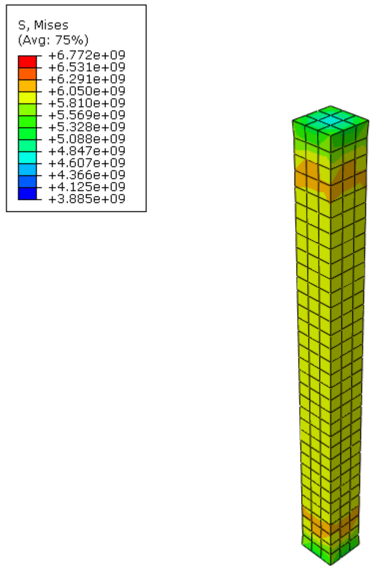


Figure L1a–13. Mises stress contour plot.

3. From the main menu bar, select **Tools→XY Data→Create.** Select **ODB field output** and click continue. In the **variables** tab, change the position to **Unique Nodal,** expand the Reaction Force 'RF' variable and check **RF2**, then expand the spatial displacement 'U' variable and check **U2**. Move on to the **Element/Nodes** tab. Choose the method selection type to be **node sets** and from the available node sets select the one which says ASSEMBLY_COUPLING_REFERENCE_POINT and click on **save**. This saves the plot of reaction force and vertical displacement versus time for the nodes of the top surface of the bar. These graphs can be found under **XYData** in the result tree and can be visualized by double clicking on them.

4. The cross sectional area of the bar which is being subjected to the tensile load = 10x10 =100 mm^2.
   And the original length of the bar = 100 mm.

   Now, stress $= \frac{\text{force}}{\text{area}}$, and strain $= \frac{\text{change in length}}{\text{original length}}$
   In order to plot stress vs strain, you first need to determine the values of stress and strain from the data you have.

5. From the main menu bar, select **Tools→XY Data → Create.** Select **Operate on XY Data** and click continue. In the operate on XY Data dialog box that appears, click on **RF2** variable that you saved previously and click on **add to expression**. Then select the / operator and manually enter a value of `100` (area of cross

section) in the expression box. Click on **save as** and name this XY Data as **Stress**. Clear the expression box by deleting the contents.

6. Similarly, in the operate on XY Data dialog box, click on **U2** variable that you saved previously and click on **add to expression**. Then select the / operator and manually enter a value of `100` (original length) in the expression box. Click on **save as** and name this XY Data as **Strain.**

7. Without closing the operate on XY Data dialog box, select the **combine** operator. Add the **Strain** XY data first by selecting it and adding it to the expression and then add the **Stress** XY Data as shown in Figure L1a-14. Click **save as** and name the XY Data as **Stress-Strain**.

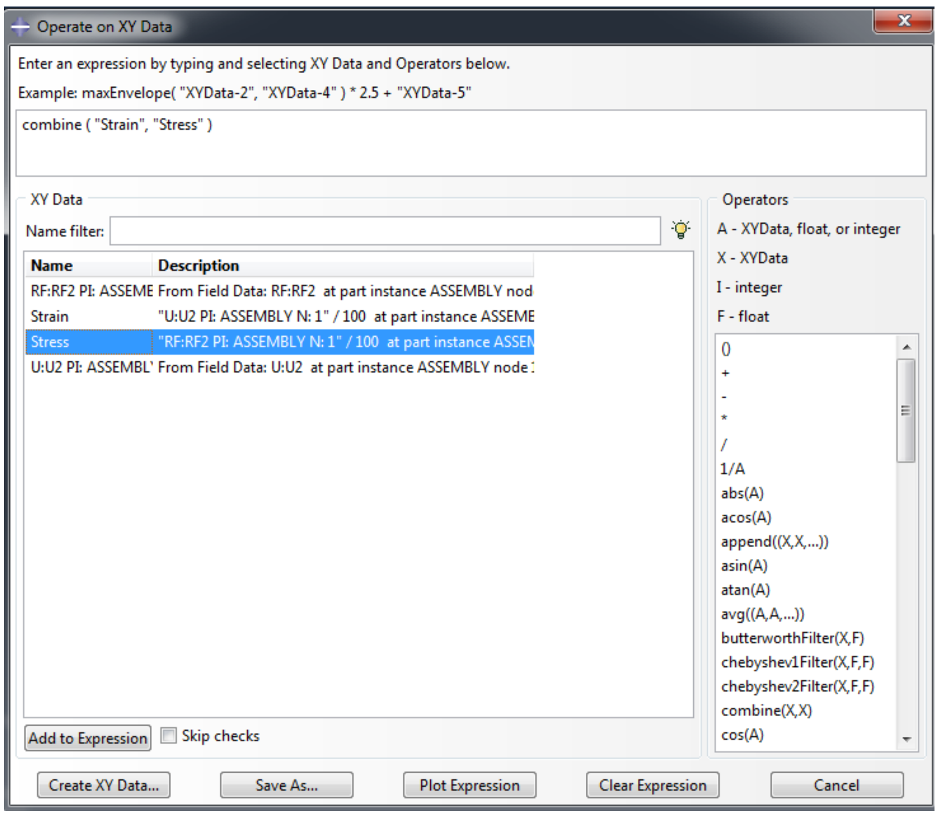


Figure L1a–14. Operations performed on XY Data to plot Stress vs. Strain

8. You can view the stress-strain plot by double clicking on it. Open the stress-strain plot. We know that the slope of the stress-strain graph represents the Young's Modulus for the material. In order to determine the slope for this plot, we can query any two points on the graph using the query tool  and find their slope using the following formula :

$$\text{Slope} = \frac{y_2 - y_1}{x_2 - x_1}$$

This slope is equal to the young’s modulus that we fed to the analysis during the material definition stage.

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