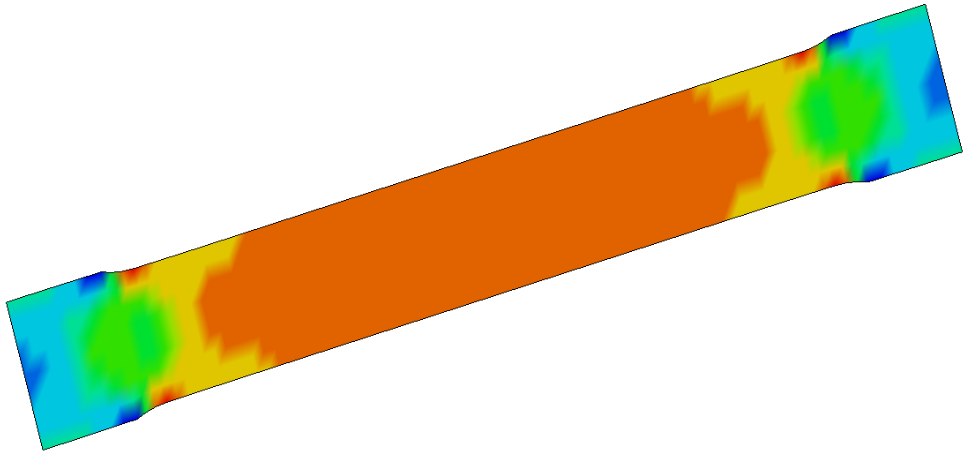
Basic Tutorials

LS-DYNA / LS-PrePost

Ex. 2. Tensile test



Contents

1 Introduction

1.1 Purpose

- Get better knowledge in basic material modeling.
- Learn how to set up explicit and implicit simulations in LS-DYNA.

1.2 Prerequisites

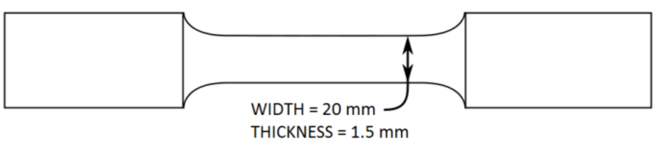
- Basic knowledge in the finite element method.
- Understand the steps in tutorial **1 - Getting Started.**

1.3 Problem Description

The task is to perform a tensile test on a flat specimen. The end of the specimen is constrained while a prescribed motion is applied on the other end. The tutorial consists of two parts.

- Perform a tensile test with the explicit solver in LS-DYNA
- Solve the problem using the implicit solver in LS-DYNA.

The task is to compare the stress vs. strain curve in an element when the uniaxial tension test is simulated using either the explicit or implicit solver in LS-DYNA.



Material properties	
Density, ρ	7850 kg/m3
Young's modulus, E	210 GPa
Poisson's Ratio, ν	0.3
Yield limit	250 MPa
Tangent modulus	1000 MPa

1.4 Data files

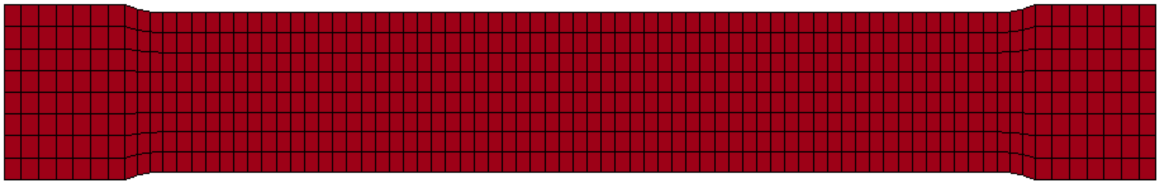
- The geometry that will be used - **tensile_test.k**.
- The final results - **tensile_test_results.k** and **tensile_test_results_implicit.k**.

2 Part A - Explicit analysis

The first part will show how to perform a tensile test with the explicit solver in LS-DYNA.

2.1 Read geometry

Open **tensile_test.k** in LS-PrePost, which contains the geometry of the test specimen.



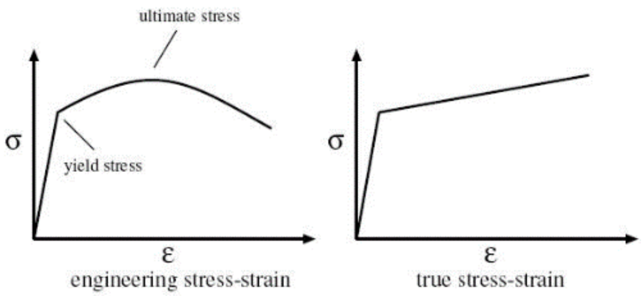
2.2 Material properties

LS-DYNA expects, for most materials, input in terms of true stress vs. true strain, not engineering stress vs. engineering strain. Normally an experimental uniaxial tension test is performed and engineering stress and strain data are obtained. Before you use this data as input in a material model in LS-DYNA, it must be converted to true stress and strain. The curves normally behave as in the figure. How you convert engineering stress and strain to true stress and strain are showed below.

Engineering stress and strain

$$\varepsilon_{eng} = \frac{L - L_0}{L_0}$$

$$\sigma_{eng} = \frac{F}{A_0}$$



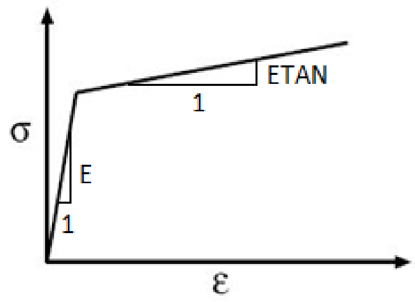
True stress and strain

$$\varepsilon_{true} = \int_{L_0}^{L} \frac{dL}{L} = ln\left(\frac{L}{L_0}\right) = ln\left(1 + \varepsilon_{eng}\right)$$

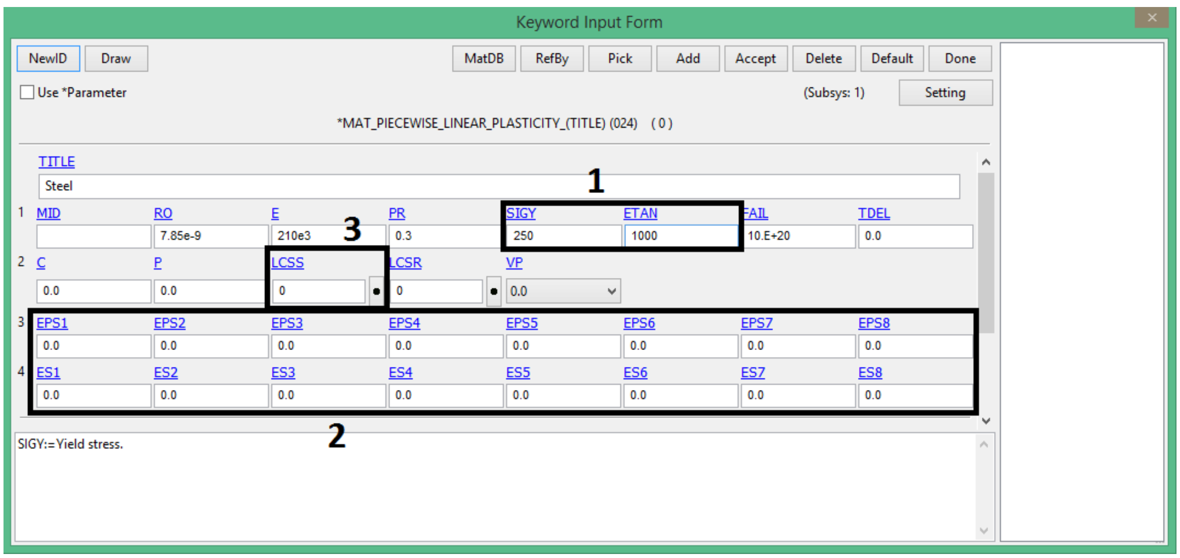
$$\sigma_{true} = \frac{F}{A} = \frac{F}{A}\frac{A_0}{A_0} = \frac{F}{A_0}\frac{A_0}{A} = \left(constant\ volume \frac{L}{L_0} = \frac{A_0}{A}\right) = \sigma_{eng}\left(1 + \varepsilon_{eng}\right)$$

To create the material card do as follows: Click **Model > Keywrd**. Activate **All** in the Keyword Manager. Double-click **MAT > 024-PIECEWISE_LINEAR_PLASTICITY**, which is an elasto-plastic material. This is a widely-used material model and the user can define the plastic behavior in several ways.

1. Define Yield stress **SIGY** and Tangent Modulus **ETAN** (see figure), which gives linear hardening.
2. Define effective plastic strain **EPS** and the corresponding yield stress value **ES**. At least two points needs to be defined. This will give a piecewise linear plastic behavior.
3. Define a curve of effective stress vs. effective plastic strain.



In this tutorial, the **first** alternative will be used to define the plastic region. Enter the title and the values as in the figure below and click **Accept**, then **Done**.



2.3 Element properties

To set the element formulation and properties do as follows:

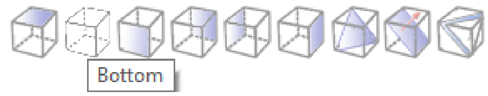
- Double click **SECTION > SHELL** in the **Keyword Manager**.
- Enter **Specimen_shell** as **TITLE**.
- Set **T1** to **1.5** and press **Enter**, **T2-T4** will then be changed to **1.5** as well and this defines the thickness of the nodes in every element.
- Click **Accept**, then **Done**.

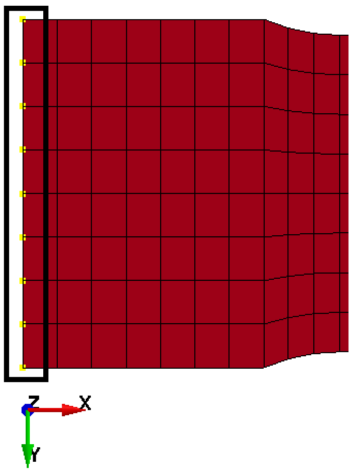
Now go to **Part** in the Keyword Manager and assign the newly created material and section. There one can also change the name of the part, to **Tensile specimen** for example**.** Click **Accept,** then **Done**.

2.4 Boundary conditions

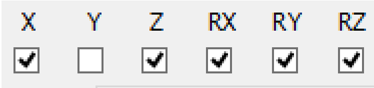
Apply the fixed boundary conditions as follows:

- Click on the **Bottom** view in the Floating Toolbar.



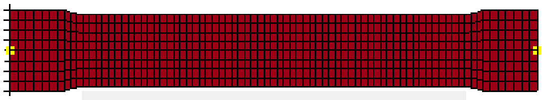


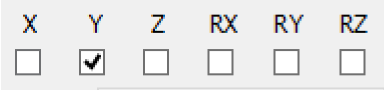
- Click **Model > CreEnt**.
- Double-click **Boundary > Spc**, click **Cre**. Make sure that **Set** is selected in the **Entity Creation** window.
- Select **Area** in the node selection window (Sel.Node). Make a box around the nine nodes as in the figure.
- Fix the nodes in X- and Z-translation and all rotations. This is done by activating **X**, **Z**, **RX**, **RY** and **RZ**.
- Click **Apply**.



Now create another boundary condition:

- In the node selection window, write **80** in the ID box and press **Enter**. Write **87** and press **Enter**. Two nodes will now be selected at the outer edges of the test specimen. Fix the nodes in Y translation by only activating **Y**.
- Click **Apply**.
- Close **Entity Creation** window.



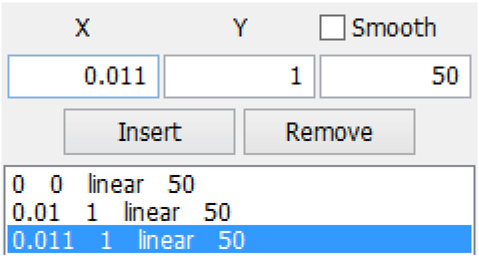


2.5 Prescribed motion/displacement

As mentioned in the first tutorial, for motions and loads, a curve must be defined that states the variation of the load/displacement/velocity etc. over time.

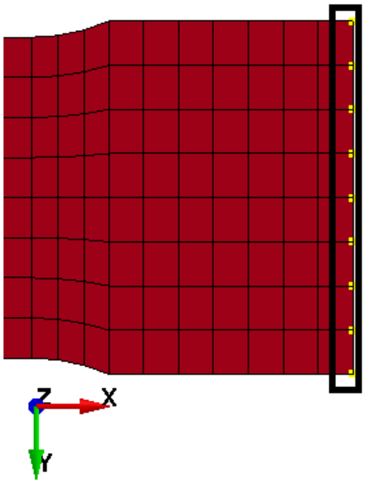
First create the curve defining the motion:

- In the top menu, click **Application > Tools > CurveGen**.
- Change **Method** to **X - Y**.
- Deactivate **Smooth**.
- Write **X = 0** and **Y = 0** (ignore the value of smooth)
- Click **Insert.** Then **0.01** and **1, Insert.**
- Finally, **0.011** and **1, Insert** (the termination time will be 0.01 s for this tutorial).
- Change **Curve Name** to e.g. **Motion**.
- Click **Create**, then **Done**.



Note that the curve could alternatively have been created using **DEFINE > CURVE** in the Keyword Manager. A curve created in CurveGen will contain a larger amount of points, compared to a curve created by **DEFINE > CURVE**.

Now apply the motion to the end of the tensile specimen:



- Click **Model > CreEnt**.
- In the **Entity Creation** window, double-click **Boundary > Prescribed Motion**.
- Select **Cre**. Change **Type** to **SET** and activate **Pick**.
- Make a box around the nine nodes as in the figure.
- Set **DOF = 1** (X-translational as degree of freedom) and **VAD = 2** (displacement as the prescribed nodal quantity, there are possibilities to prescribe the nodal velocity or acceleration as well).
- Click on **LCID** and select the previously created curve.
- Set **SF = 20**, which implies that the node set will be moved 20 mm.
- Click **Apply**, then **Done**.

2.6 Set the termination time

To set the termination time:

- Click **Model > Keywrd**.
- Double-click **CONTROL > TERMINATION**.
- Set **ENDTIM** to **0.01**. The simulation will then last for 0.01 seconds.
- Click **Accept** and then **Done**.

2.7 Output

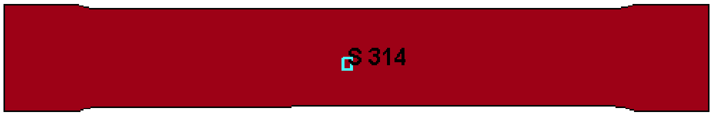
Here the output to be saved by LS-DYNA during the simulation is defined, first specify d3plot output:

- Create d3plots.
- Click **Model > Keywrd**.
- Double-click **DATABASE > BINARY_D3PLOT**.
- Set **DT = 5e-4**.
- Click **Accept** and then **Done**.

Since the task was to compare the stress vs. strain curve for an element with the material curve, we want to obtain this data frequently. In the first tutorial (1 – Getting Started) we used the **History** command to plot information. The time-history graphs created by **History,** use information found in the d3plot files. To obtain data more frequently we can specify which type of data that are of interest and how often it will be printed for selected elements. Do as follows:

- Click **Model > Keywrd**.
- Double-click **DATABASE > ASCII_option**. We are interested in the stress and strains in the elements, therefore activate **ELOUT**. On the same row, set **DT = 5e-6**.
- Click **Accept**, then **Done**.

The element/elements that we want to gather data from must be defined:



- Double-click **DATABASE > HISTORY_SHELL** in the **Keyword Manager**.
- Write, for example, shell element number **314** under **ID1**, which is an element close to the center of the specimen.
- Click **Insert, Accept,** then **Done.**

Check the model to see that everything looks okay.

2.8 Save

The model is now ready to be saved. **File > Save As > Save Keyword As**. Choose a folder path and name your file **tensile_test_model.k** for example. **Note** that the folder path cannot contain any spaces. Close **LS-PrePost**.

2.9 Run the simulation

Run the simulation using **LS-Run**, see exercise **1. Getting Started** for more information on how to do this. The simulation runs in less than a minute to completion. Use the “SMP Single” preset.

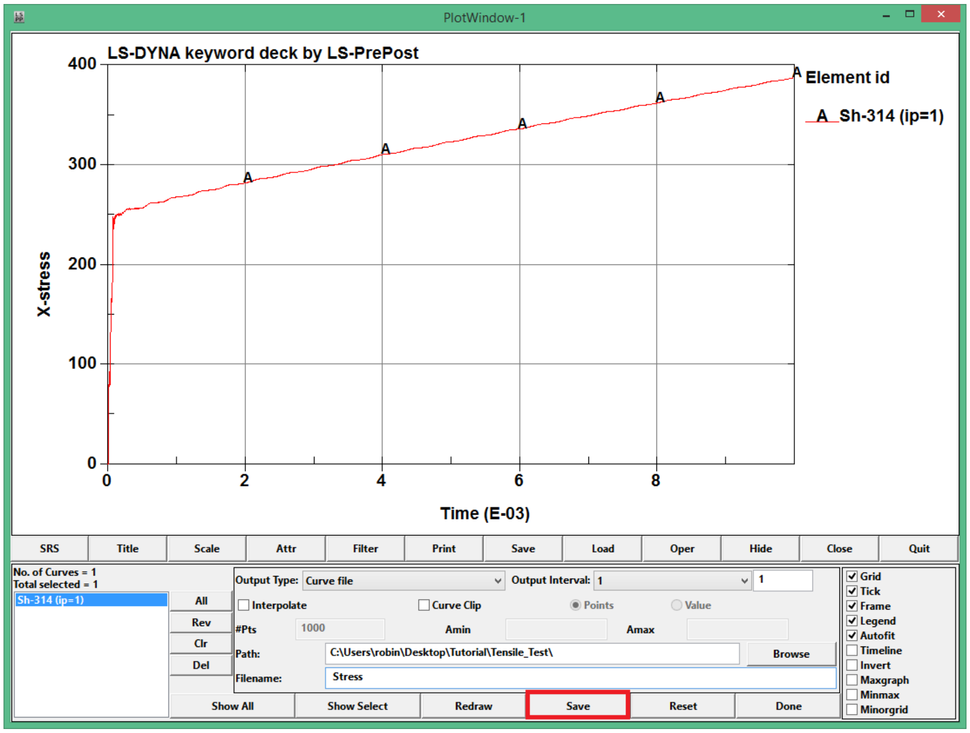
2.10 Post processing

Open the **d3plot** in LS-PrePost from **LS-Run** or from the File menu of LS-PrePost.

Start to press the **Forward** button in **Animate** toolbar to see what happens with the specimen.

Now plot the stress-strain-curve from the elout file, first plot and save stress vs. time:

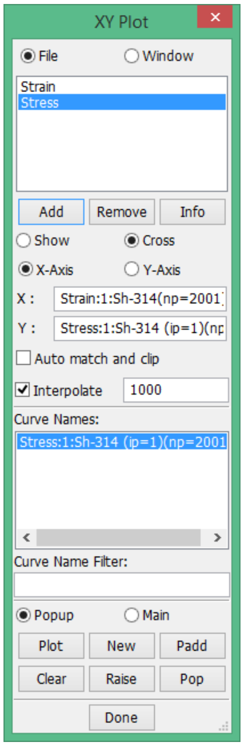
- Click **Post > ASCII**.
- Select **elout*** and click **Load**.
- Select **Sh-314**, **Ip-1** or **Ip-2** (doesn't matter which integration point in this case) and **9-Effective Stress (v-m)**.
- Click **Plot.**
- In the **PlotWindow,** click **Save.**
- Let the **Output Type** be **Curve file**.
- Enter **Stress** as the **Filename** (also set correct folder path).
- Click **Save** (located at the bottom toolbar, see red square).
- Click **Quit**



Now plot and save the effective plastic strain vs. time:

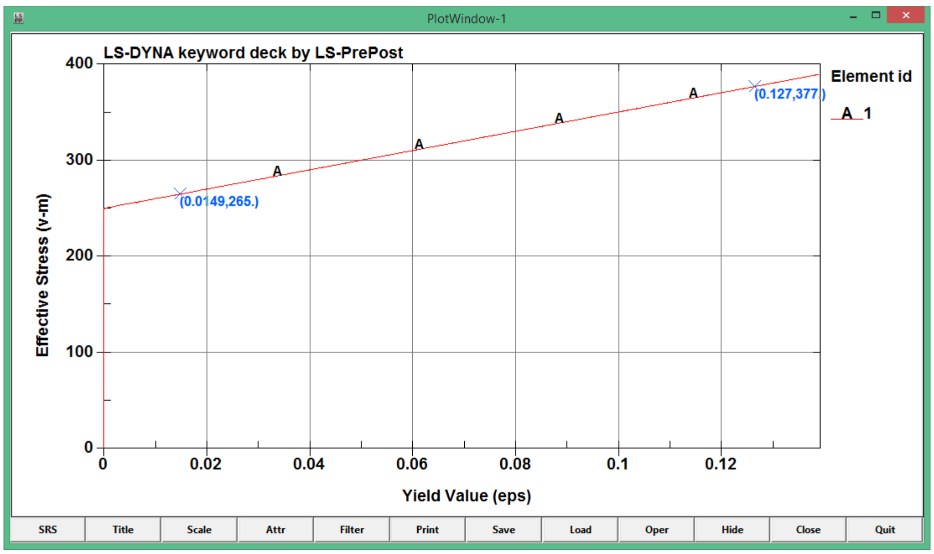
- Select **7-Yield Value (eps)** instead of **Effective Stress (v-m)**.
- Click **Plot**.
- Save the curve in the same way as for the stress, but name the curve **Strain**.
- Close **PlotWindow** and **ASCII**

Now combine the above two curves (Stress and Strain) to plot the stress vs. strain:



- Click **Post > XYPlot**.
- Activate **Cross.**
- Note that **X-Axis** is activated, select **Strain** in the top box and it will pop-up under **Curve Names**.
- Click on **Strain:1:Sh-314…..** and it will be inserted in **X:**.
- When you do this, **Y-Axis** will be activated.
- Select **Stress** in the top box and click on **Stress:1:Sh-314…** under **Curve Names** and it will be inserted in **Y:**.
- Click **Plot**.

The curve shows effective plastic strain vs. effective stress (von Mises). We see that the yield limit is about 250 MPa as we stated in the material model.



Let's check that the tangent modulus **ETAN** (1000 MPa) are correct as well. Click on two points on the curve and calculate the tangent. Using the points in the figure gives:

$$\frac{377\ MPa - 265\ MPa}{0.127 - 0.0149} = 999\ MPa$$

Finally save the curve and name it **Explicit_curve**. Part A is now completed and you may close LS-PrePost.

3 Part B - Implicit analysis

Run the tensile test using the implicit time-step method instead of using explicit time-step method used in Part A.

Open the file **tensile_test_model.k** that you created earlier in LS-PrePost. Note: This can be done also from **LS-Run**. Do as follows to change to the implicit time-step method:

- Click **Model > Keywrd**.
- In the Keyword Manager, make sure that **All** is selected, double-click **CONTROL > IMPLICIT_GENERAL**. This keyword is mandatory to activate the implicit time-stepping method.
- Set **IMFLAG = 1**, which implies an implicit analysis. As default in LS-DYNA, a non-linear, static simulation will be performed when just defining an implicit solution. If you want to perform a dynamic implicit simulation for example, you will have to specify that using other control cards.
- Set the initial time step **DT0 = 5e-4**.
- Set **IGS = 1**, which can improve convergence in tensile.
- Click **Accept**, then **Done**.

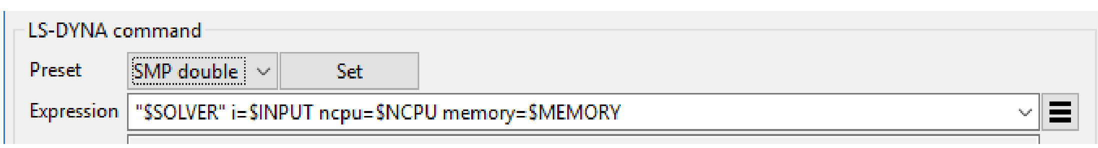
3.1 Save the model

Save your model as **tensile_test_model_implicit.k.** Close LS-PrePost.

3.2 Run the simulation

Implicit analyses should always be run using the double precision version of LS-DYNA, as the implicit solver is more sensitive to round-off errors.

Run the simulation using **LS-Run**, see exercise **1. Getting Started** for more information on how to do this. The simulation runs in less than a minute to completion. To use the double precision version, make sure to use the “SMP double” preset in LS-Run, see the image below.



3.3 Post processing

Open the **d3plot** in LS-PrePost. Start to press the **Forward** button in **Animate** toolbar to see what happens with the specimen.

The time step in this implicit analysis was set to **5e-4** s, which is larger than the time between the output states in **ELOUT** (5e-6). Therefore, elout data will only be obtained every 5e-4 second.

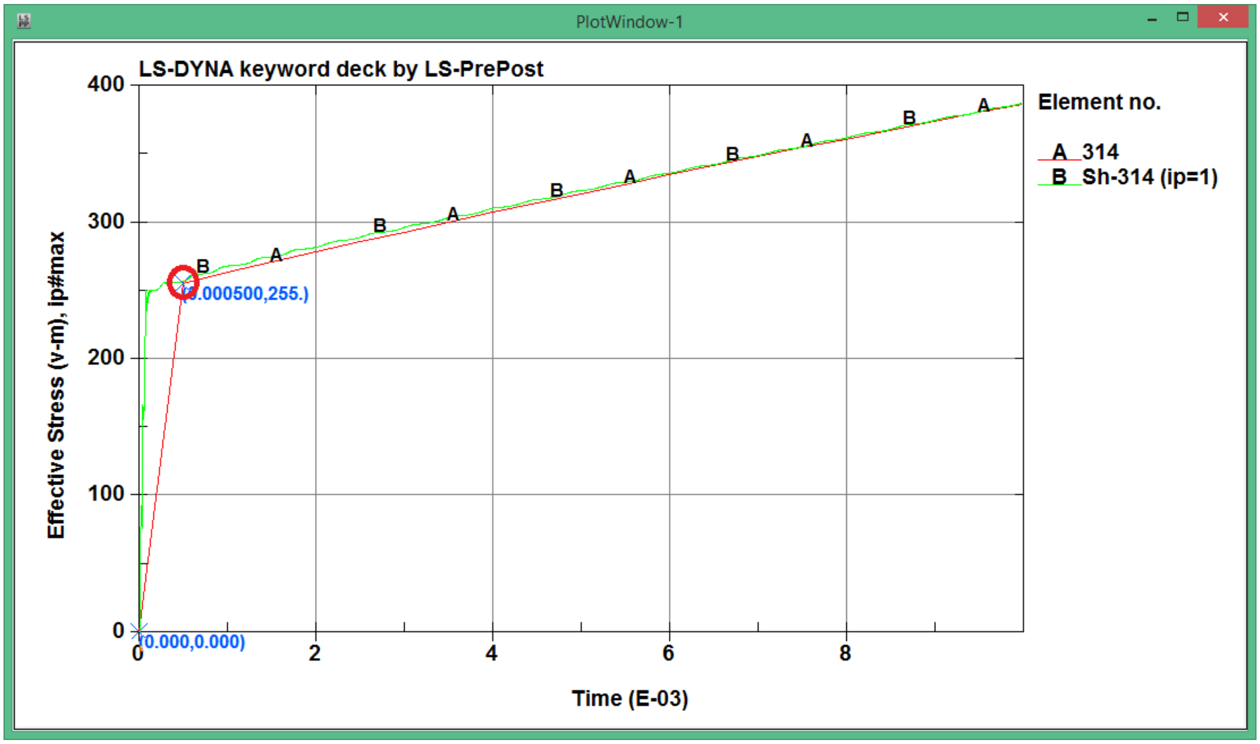
Plot the **Effective stress (v-m)** from **Post > ASCII > elout*** as for Part A of the exercise.

Save the plot, name it **Stress_imp**.

The first time we obtain any data is after 5e-4 seconds (if initial state is ignored), visualized with a red circle in the figure. We will therefore have no information about what happens between **t=0** and **t=5e-4**.

Now compare the results with the result from the explicit analysis:

- Click **Load > Add File (XY Data)** and find the curve file **Stress** that we created earlier.
- Select **Stress** and click **Select**.
- Note the difference at which time we reach the yield limit for the two curves. This is an effect of that the explicit analysis uses a smaller time step size.
- Close the **PlotWindow**.

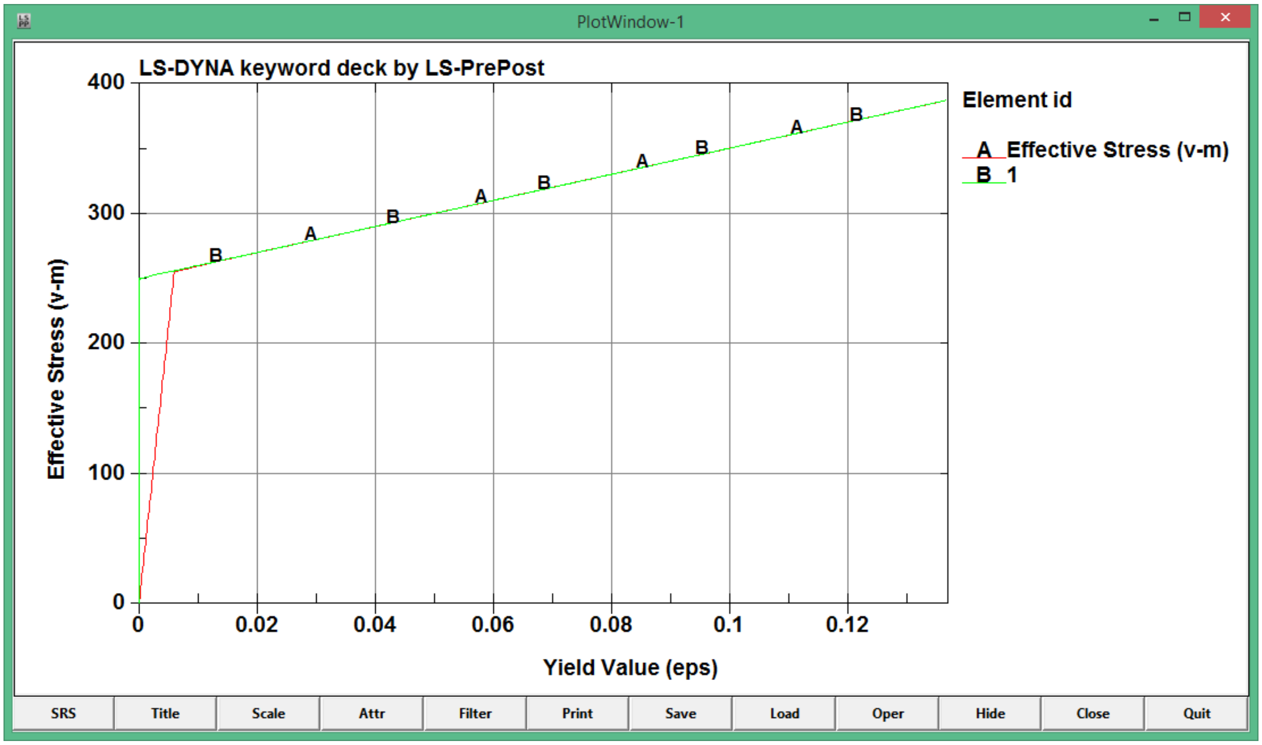


Now, plot and save the strain vs. time curve:

- Plot the **Yield Value (eps)** and save the curve to **Strain_imp**.
- Close **PlotWindow** and **ASCII**.

Finally create the stress vs. strain curve for the element and compare it that from the explicit analysis:

- Create an XY-plot of **Strain_imp** and **Stress_imp**.
- Click Load > Add File (XY Data) and add the curve Explicit_curve in the PlotWindow.
- Select Explicit_curve and click Select.
- We now know that the curves differ in the beginning of the simulation due to the time step.
- After the first time step, the curves are very similar, this is because we use linear hardening in the material model.



4 Comments

In this exercise, you have performed a simulation of a tensile test both with explicit and implicit time integration. It is always up to the user to decide whether the results are good enough. An experimental tensile test is a slow process and can be considered as quasi-static i.e. not affected by mass inertia. How much can the load speed be scaled while still maintaining minimal mass inertia effects? How much kinetic energy can be accepted allowed in the analysis? It is difficult to give a general answer and always up to the user to decide.

4.1 Explicit versus implicit

Explicit schemes give the configuration at time t_{n+1} as an explicit function of earlier configurations.

$$x_{n+1} = f(x_n, x_{n-1}, \dots, t_{n+1}, t_n, \dots)$$

Implicit schemes give the configuration at time t_{n+1} as an implicit function of the unknown as well as earlier configurations, i.e. solves an equation system.

$$x_{n+1} = f(x_{n+1}, x_n, x_{n-1}, \dots, t_{n+1}, t_n, \dots)$$

Explicit

- Computationally more efficient since nodal accelerations are solved directly and not iteratively
- Conditionally stable, requires small time steps.
- Memory efficient
- Good for dynamic problems, e.g. impact and shock problems

Implicit

- Solves an equation system at every time step
- Cost of every time step is unknown, due to variations in convergence behavior
- Unconditionally stable for linear problems, which means that there is no limit to the size of the time increment
- Implicit time steps are generally several orders of magnitude larger than explicit time steps.

5 Optional exercises

4. Decrease the initial time step size in **CONTROL_IMPLICIT_GENERAL** to see how this affects the results.
5. Refine the mesh. (**EleTol > EleEdit > Split/Merge**). Does this affect the results? (If the simulation time becomes too long, decrease the termination time)