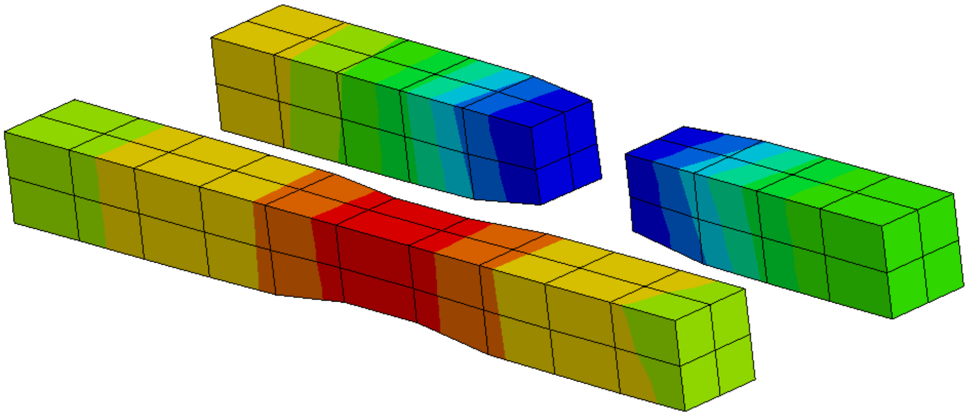
Basic Tutorials

LS-DYNA / LS-PrePost

Ex. 5. Hardening and failure



Contents

1 Introduction

1.1 Purpose

- Get familiar with kinematic and isotropic hardening.
- Get better knowledge in failure criterion in LS-DYNA.

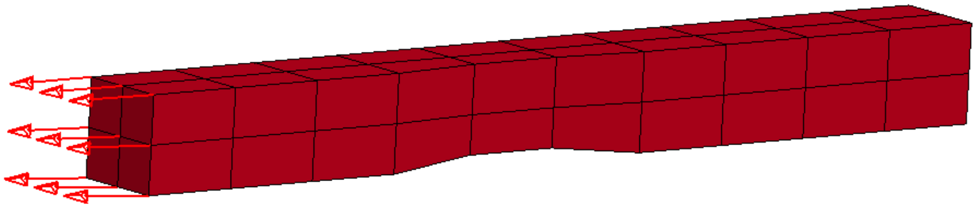
1.2 Prerequisites

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

1.3 Problem Description

A beam is subjected to a cyclic motion in one end and fixed in the other end. The dimension of the beam is 100x10x10 mm, with a slight modification in the middle, to have a weak section where the major part of the deformation will occur. Two similar beams will be analyzed, one with isotropic hardening and one with kinematic hardening.

Material properties	
Density, ρ	7850 kg/m3
Young's modulus, E	210 GPa
Poisson's Ratio, ν	0.3
Yield limit	400 MPa
Tangent modulus	1000 MPa

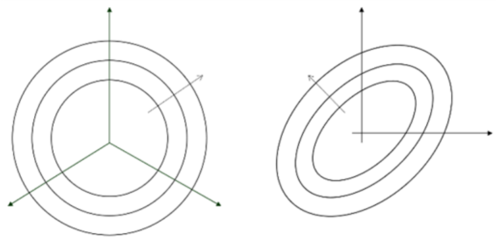


1.4 Theory – Hardening

The image below illustrates the difference between kinematic and isotropic hardening.

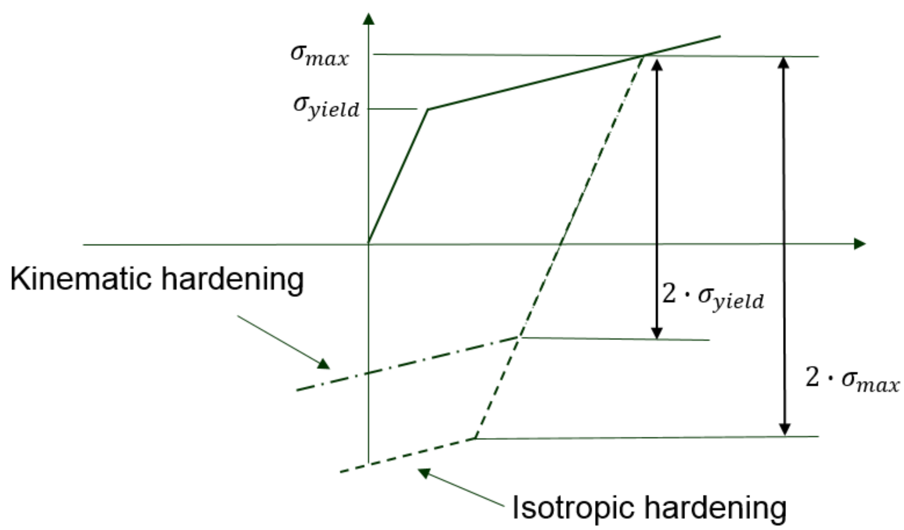
Isotropic

The yield surface expands with increasing plastic strain but does not change in shape.



Kinematic

The size of the yield surface does not change, it is translated in the direction of plastic strain. This is necessary to treat cyclic deformation.



1.5 Data files

The input file for the exercise containing all necessary keywords except material and output keywords is **hardening.k.** The solution to the exercise can be found in the file **hardening_results.k**.

2 Create the model

Open **hardening.k** in LS-PrePost, it contains all necessary keywords except the material and output keywords.

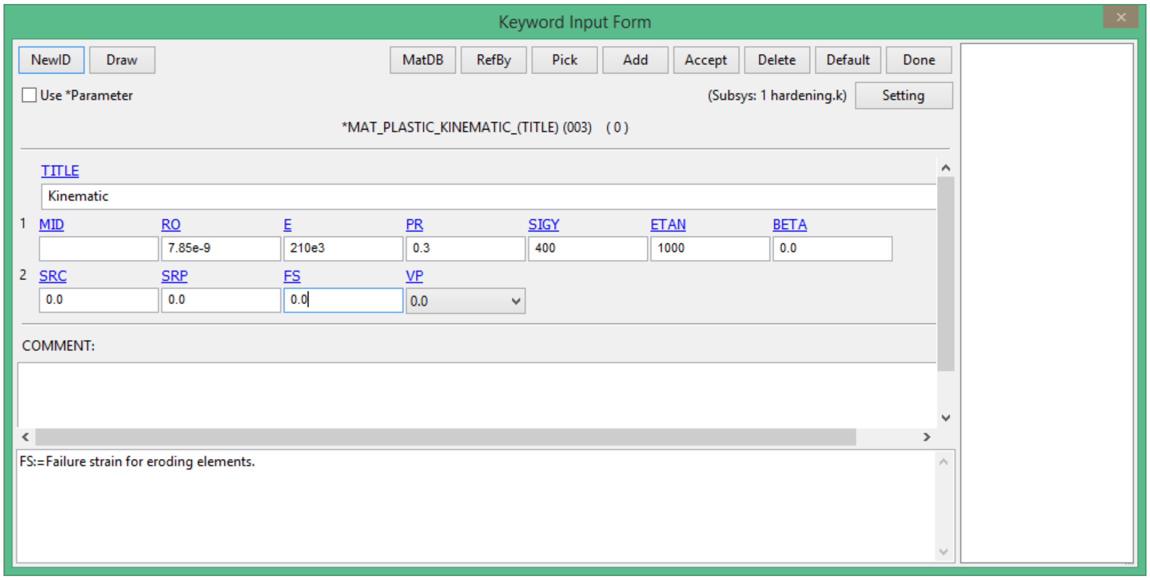
Have a look at the keywords in the model. The cyclic motion is applied with the keyword **BOUNDARY_PRESCRIBED_MOTION_SET**, using the curve which is defined with **DEFINE_CURVE**.
The other side of the beam is constrained using **BOUNDARY_SPC_SET**.

By observing the keywords using the Keyword Manager, one can see that the material card needs to be defined.

2.1 Material

Create the material card as follows:

- Double-click **MAT > 003-PLASTIC_KINEMATIC** in **Keyword Manager**.
- Enter the values from **RO – ETAN** as described in the problem description.
- **BETA** is the hardening parameter and can vary between 0 and 1. Kinematic hardening is obtained by setting **BETA** to **0** and isotropic hardening by setting **BETA** to **1**.
- Name the material **Kinematic** and set **BETA** to **0**.
- Click **Accept**, then click **NewID**.
    - Note: The defined parameter values will be kept and only the ID will be updated to max ID + 1 when using **NewID** functionality in LS-PrePost.



Now create also an isotropic material model:

- Change **BETA** to **1** and name the material **Isotropic**.
- Click **Accept**.
- Click **Done**.

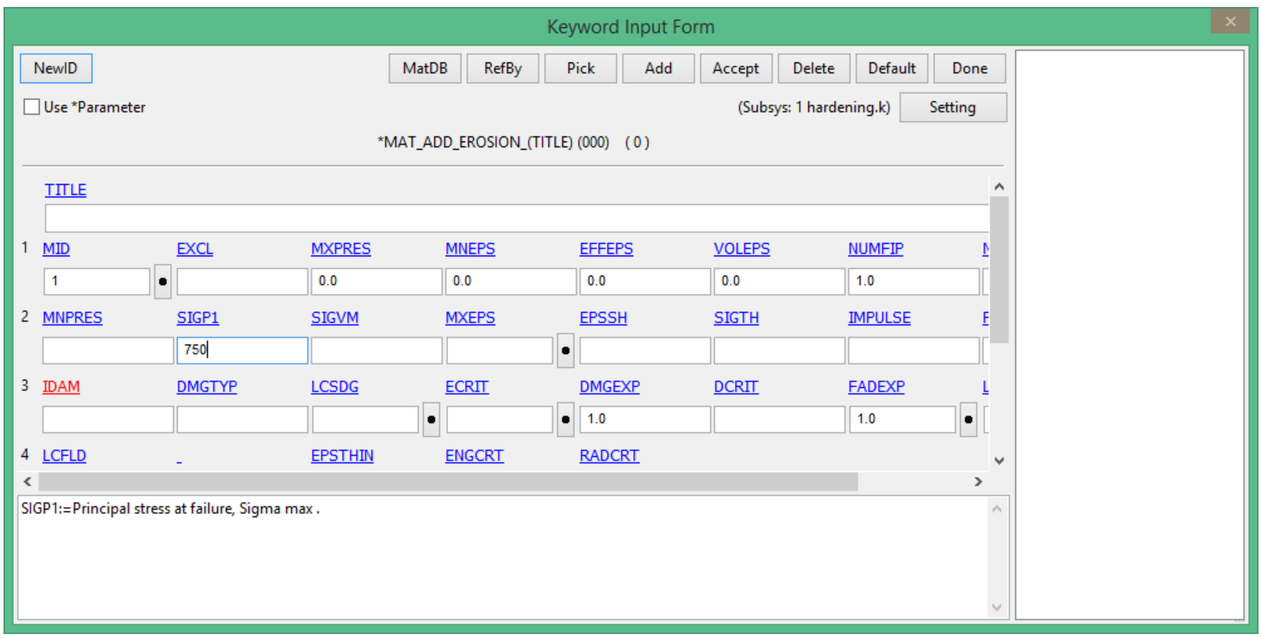
Failure will also be considered in this model. The parameter **FS** could be used to set an effective plastic strain when the elements will erode. In this tutorial, we will instead define a stress level that specifies when failure will occur. This will be done through using another keyword.

Connect the materials to the 2 parts:

- Go to **Part > Part** in **Keyword Manager** and define the newly created materials to the different beams.
- Kinetic hardening to part one (Red) and isotropic hardening to part two (Blue).

Failure will be set with a special keyword as follows:

- Double-click **MAT > 000-ADD_EROSION** in the Keyword Manager**.** This "add erosion" functionality can be added to many material models in LS-DYNA. This keyword contains different parameters that can define when failure will occur, we will in this case use the maximum principal stress **SIGP1**.
- Set **MID** to **1**, which is the same material ID as for the kinematic material model. LS-DYNA will interpret that this instance of ***MAT_ADD_EROSION** should be connected to the previously defined material model with kinematic hardening. Set **SIGP1** to **750**, click **Accept**.
- Click **NewID** and change **MID** to **2**, click **Accept** and **Done**.

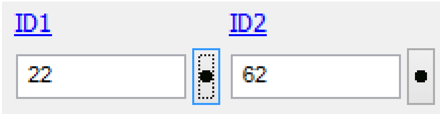


2.2 Output

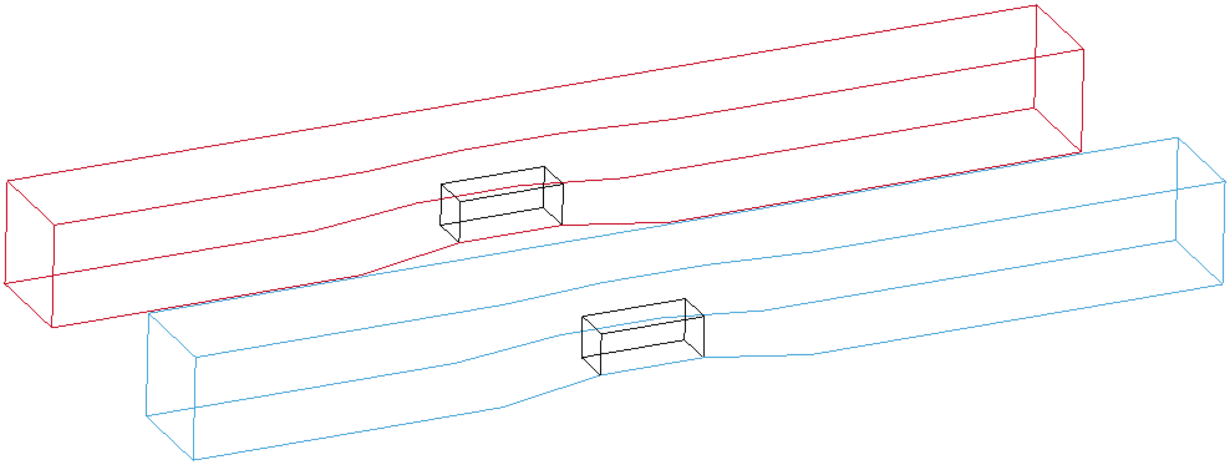
Define ASCII output databases:

- Double-click **DATABASE > ASCII_option**.
- Set **Default DT = 5e-5** and press **Enter**.
- We are interested in the stress and strains in the elements, therefore activate **ELOUT**.
- Also activate **BNDOUT**, which will give information about the force that is needed to perform the motion for each node in the boundary.
- Click **Accept**, then **Done**.

We also have to define for which elements data will be saved in the file **ELOUT:**



- Double-click **DATABASE > HISTORY_SOLID** in the **Keyword Manager**.
- Write **22** under **ID1** and **62** under **ID2**. These are two elements in the thinner part of the cross-section.
- Click **Insert**, then **Accept** and **Done**.
    - Note: Click **Model > Display > Database > History Solid > All** in the **Entity Selection** to highlight element **22** and **62**. To hide them again, click **None**.



ELOUT will not give any information about the strains as default. A keyword must therefore be added:

- Double-click **DATABASE > EXTENT_BINARY** in the **Keyword Manager**.
- Set **STRFLG** to **1**. LS-DYNA will then write strain tensor data to **d3plot** and **ELOUT**.
- Click **Accept**, then **Done**.

Save the keyword file and start the simulation using the single precision SMP version of LS-DYNA.

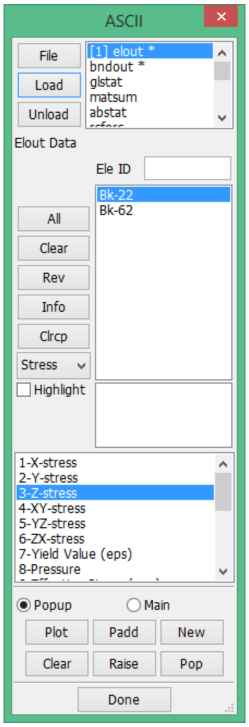
2.3 Post-processing

During the simulation, we get information in the **messag** and **d3hsp** files about that solid elements 61 to 64 failed at a certain time, due to our failure criteria.

```
40000 t 2.2165E-02 dt 5.44E-07 flush i/o buffers
41478 t 2.3000E-02 dt 5.40E-07 write d3plot file
  solid element           62  failed at time  2.3129E-02
  solid element           61  failed at time  2.3129E-02
  solid element           64  failed at time  2.3129E-02
  solid element           63  failed at time  2.3130E-02
43186 t 2.4000E-02 dt 5.83E-07 write d3plot file
44919 t 2.5000E-02 dt 5.73E-07 write d3plot file
```

Open **d3plot** with LS-PrePost. Use the **Animate Toolbar** to see what happens.

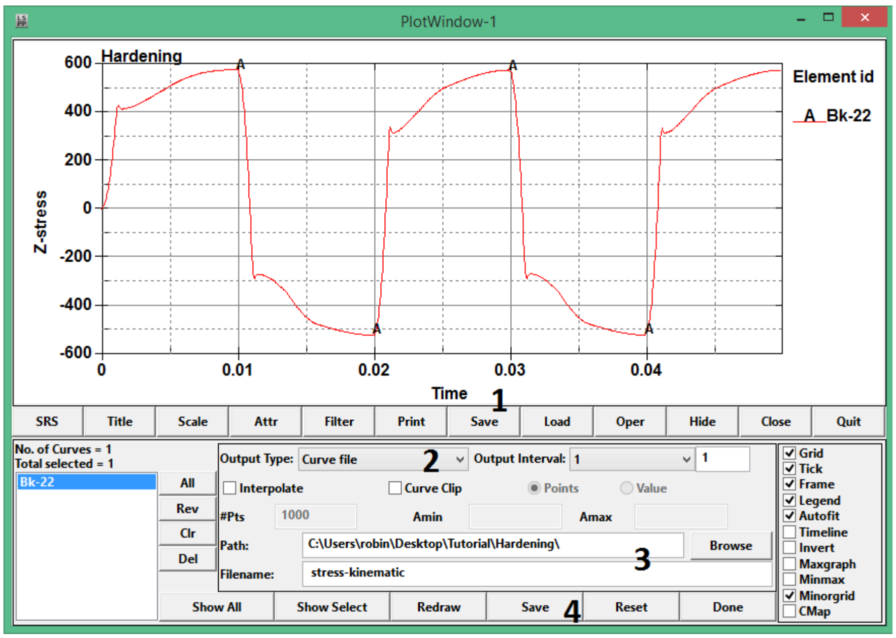
Four elements in the blue part, that uses isotropic hardening, will fail. Plot the stresses in the elements with data save to the **ELOUT** file:

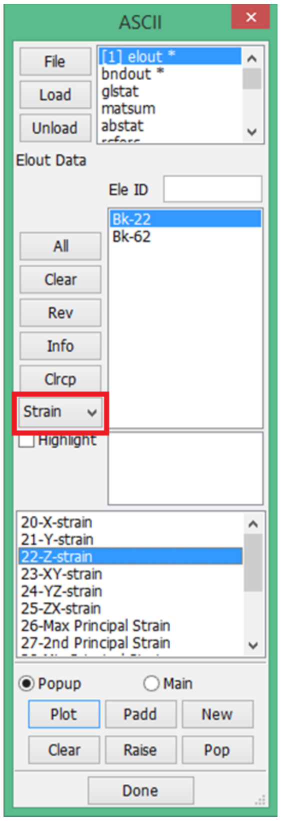


- Click **Post > ASCII**.
- Select **elout*** and click **Load**.
- Select **Bk-22** and **Z-stress**.
- Click **Plot**.

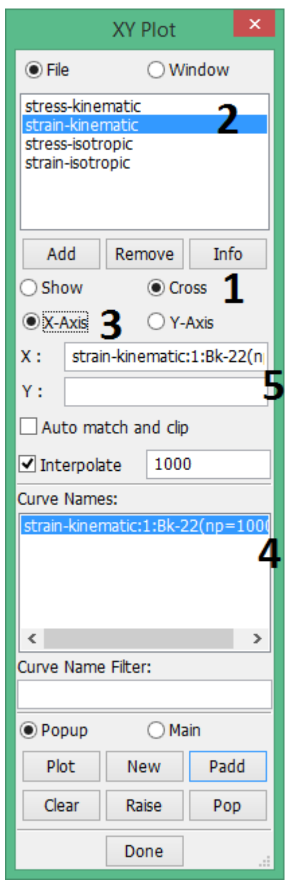
In the PlotWindow:

- Click **Save**. (1)
- Set **Output Type** to **Curve file**. (2)
- Write **stress-kinematic** as **Filename**. (3)
- Click on the **Save** button to save the curve. (4)
- Close PlotWindow.





- In the **ASCII** window, change **Stress** to **Strain** (red box in figure).
- Select **Bk-22** and **Z-strain**.
- Click **Plot**.
- Save the curve, name it **strain-kinematic**.
- Perform the previous steps again to save the curves **Z-stress** and **Z-strain** for element **62 (Bk-62)**. Name them **stress-isotropic** and **strain-isotropic**.

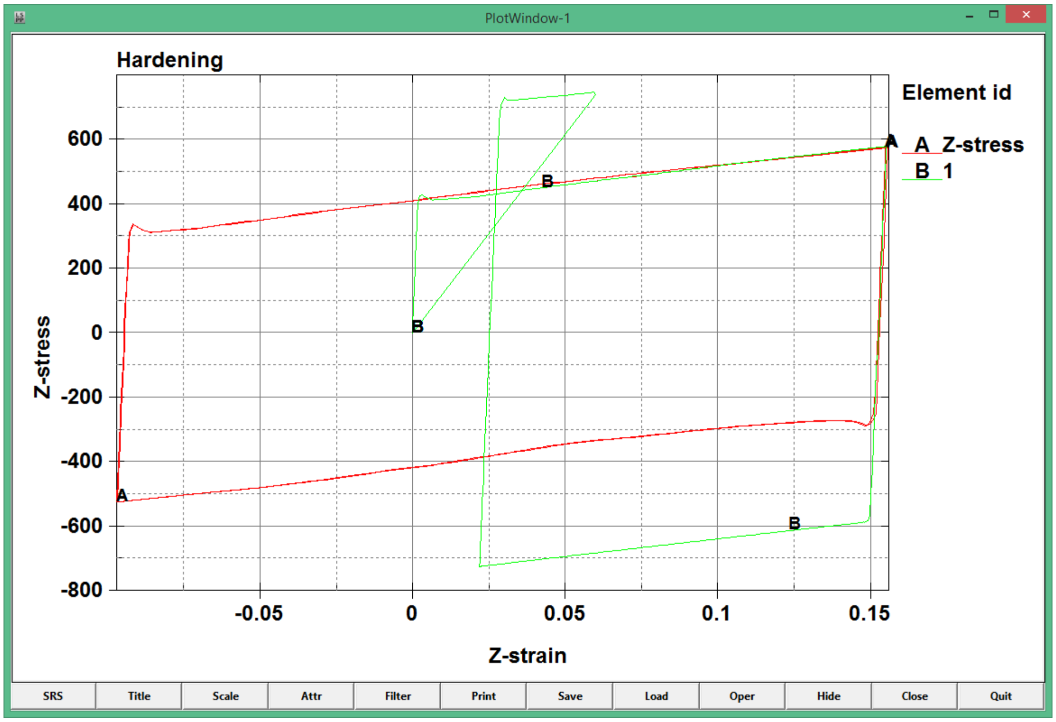


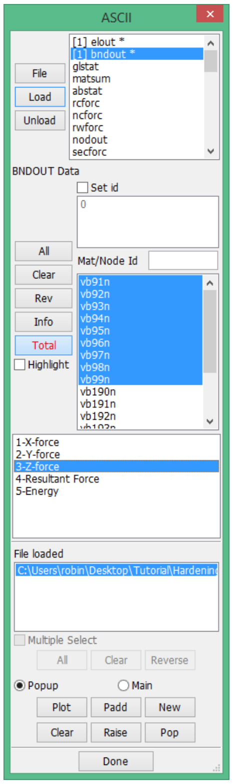
Now create the stress vs strain curves using the cross-plot feature of LS-PrePost:

- Click **Post** > **XYPlot**.
- Select **Cross**. (1)
- Select **strain-kinematic** and it will pop up under Curve Names. (2) Make sure **X-axis** is activated. (3)
- Click on **strain-kinematic** under **Curve Names**. (4) Strain-kinematic will then pop-up next to **X:**. (5)
- In a similar manner, set **stress-kinematic** to **Y:**.
- Click **Plot**.

Now go back to **XY Plot** without closing **PlotWindow**. Perform the previous steps to create a XY plot for the isotropic curves and finally click **Padd** instead of **Plot**.

The red curve shows the kinematic hardening while the green curve shows isotropic hardening. Compare them with the curves in section **1.4 Theory – Hardening.** to understand the differences.



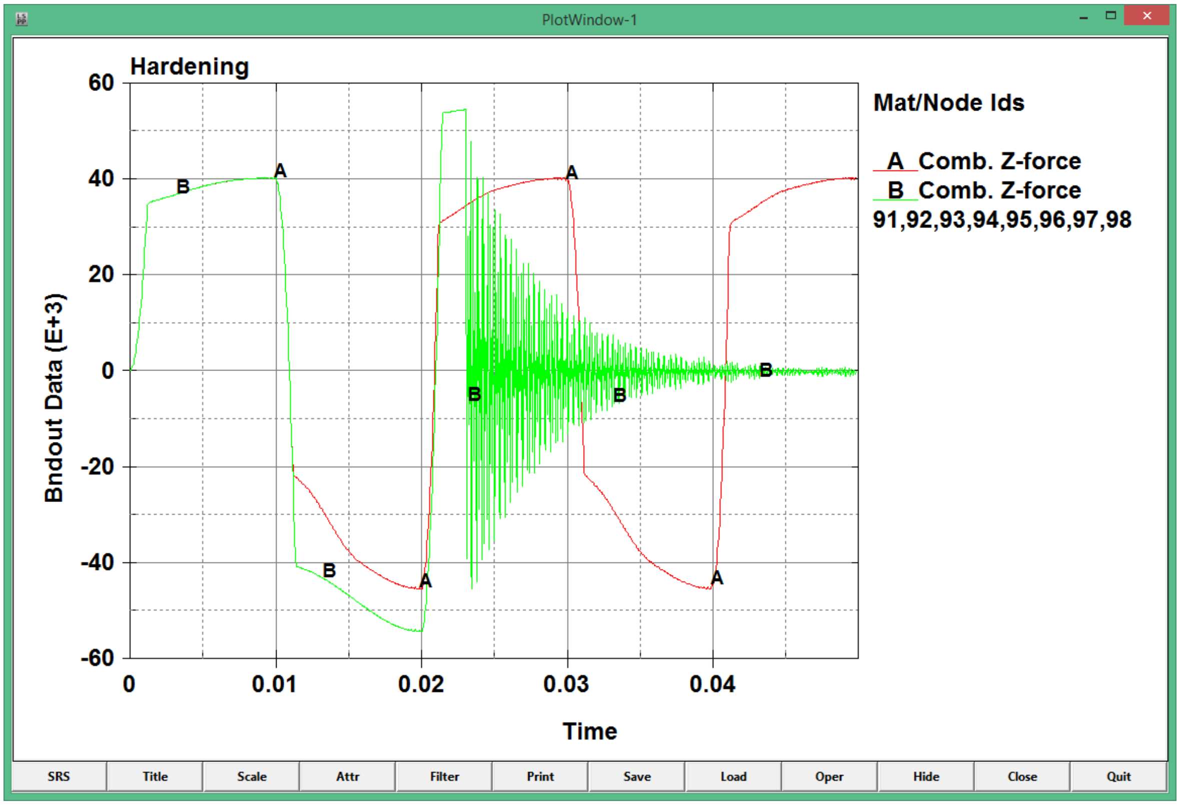


Now close **PlotWindow** and click **Post > ASCII**. Proceed to plot the applied forces:

- Select **bndout*** and click **Load**.
- Click **Total** and select **vb91n** to **vb99n** (the nodes in the boundary for the part with kinematic hardening).
- Select **Z-force** and click **Plot**.

Do the same thing for **vb190** to **vb198** and click **Padd** instead of **Plot**.

In the plot window one can now see the total Z-force required to perform the motion for the different beams.



3 Summary

The focus in this exercise was to understand how isotropic and kinematic hardening works and how you can use it in LS-DYNA. Material failure has also been demonstrated which can be often be set directly in the material model, or by adding the keyword MAT_ADD_EROSION (the latter has more failure criteria).

In this exercise, a very simple hardening curve was used. Only two parameters defined it: the tangent modulus **ETAN** and the yield stress **SIGY**. Normally, an experiment is performed to obtain a force vs displacement curve for the material. This data is converted to true stress and true strain and inserted into a material model in LS-DYNA. This curve is only valid until necking; therefore some simple tests need to be performed to obtain the correct stress vs strain behavior after this point. This is done iteratively by connecting to the experimental force vs displacement curve. In tutorial **7 – Parameter identification using LS-OPT,** this will be covered using the optimization program LS-OPT

4 Optional exercises

1. Change the value of the hardening parameter in **MAT003**. At which **BETA** will the specimen fracture?

2. Try to set another failure criterion (**EFFEPS** for example) in **MAT_ADD_EROSION** or make use of **FS** in **MAT003** to understand how they work. Multiple failure criterions can be set simultaneously.

3. What will happen if you change material to **MAT024-PIECEWISE_LINEAR_PLASTICITY**? Is it an isotropic or kinematic material model?