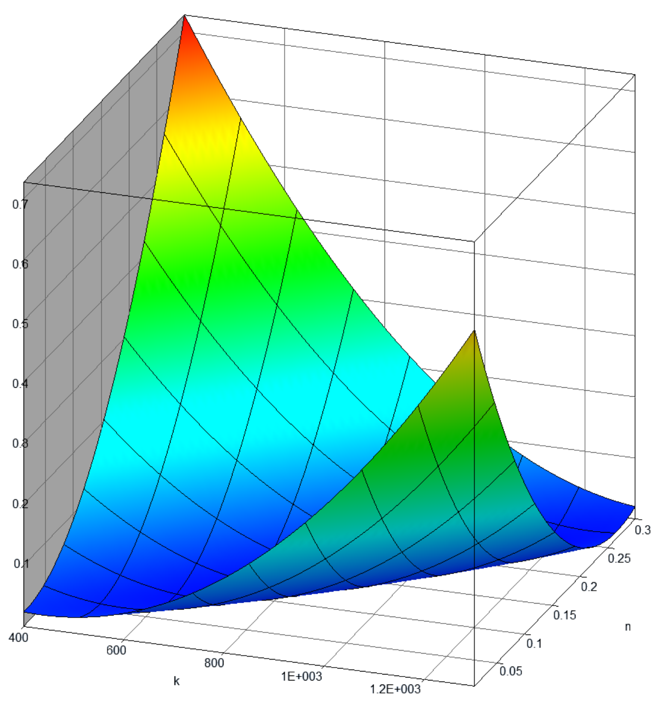
Basic Tutorials

LS-DYNA / LS-PrePost

Ex. 7. Parameter identification using LS-OPT



Contents

1 Introduction

LS-OPT is a graphical optimization tool developed by LSTC that links perfectly with LS-DYNA. It allows the user to structure the design process, explore the design space and compute optimal designs according to specified constraints and objectives. The program is also highly suited to the solution of system identification problems and stochastic analysis.

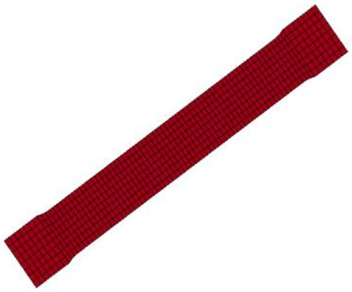
The graphical tool LS-OPTui interfaces with LS-DYNA and provides an environment to specify optimization input, monitor and control parallel simulations and post-process optimization data, as well as viewing multiple designs using LS-PrePost.

For more information about the functions in LS-OPT, see LS-OPT User's Manual.

The purpose with this tutorial is to get familiar with the LS-OPT software.

1.1 Prerequisites

- Basic knowledge in the finite element method and optimization theory.



2 Problem description - Tensile test

Use a power law material model ($\sigma_y = k\varepsilon^n$) in LS-DYNA, described by the two parameters k and n, to simulate a tensile test. These two parameters will be optimized to fit experimental tensile test data (see curve) i.e. minimize the difference between experimental and simulation force vs displacement curves in a least square sense.

Optimization formulation

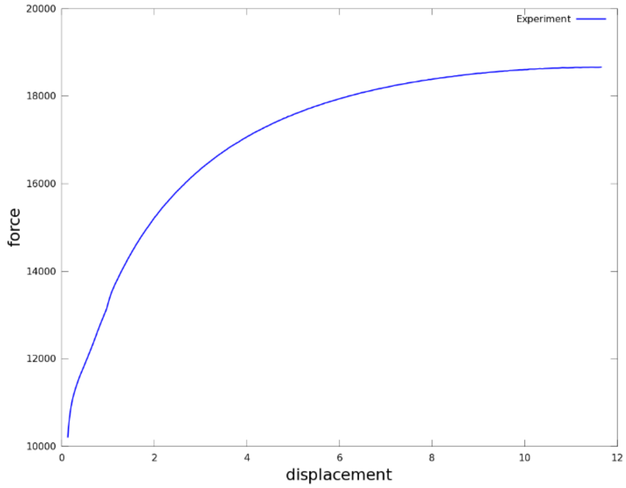
Find k, n

Minimizing $MSE = \frac{1}{P}\sum_{i=1}^{P} W_i \left(\frac{f_i(k,n) - G_i}{s_i}\right)^2$

Subject to $400 \le k \le 1300$
$0.01 \le n \le 0.3$

where
MSE: mean square error
W_i: weight factor

s_i: scale factor
P: number of experimental points
f_i: simulation values
G_i: target values



2.1 Data files

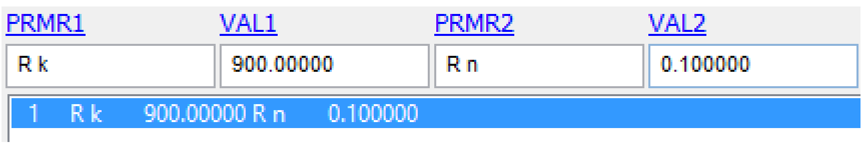
- The keyword file with the tensile specimen - **tensile_test.k**.
- The experimental force vs displacement data - **exp_force_disp.txt**.
- The final LS-OPT setup - **param_ident_results.lsopt**.

2.2 Check model

Open **tensile_test.k** in LS-PrePost.

2.3 Parameters

In LS-OPT, the parameters k and n will be optimized. We will therefore use parameters in our keyword file. Check the keyword **PARAMETER_PARAMETER**. The name of the parameter is set in **PRMRX** and the value in **VALX.** Before the name of the parameter, you have to define if the parameter is a real number (**R**) or an integer (**I**). Two parameters are defined here, **k = 900** and **n = 0.1**, both as real numbers.

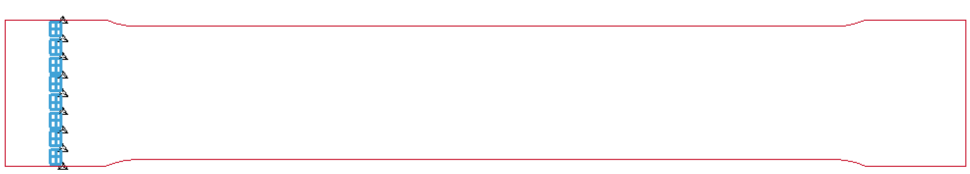


Open **MAT_POWER_LAW_PLASTICITY**. Click on **Use *Parameter** in the top left corner. LS-PrePost will then show which options that are defined using parameters and parameters can also be added in this mode. The strength coefficient (**K**) and the hardening exponent (**N**) are defined with parameters for this material. Uncheck **Use *Parameter** and click **Done** to close the window, click **Ignore** in the pop-up dialog.

2.4 Output

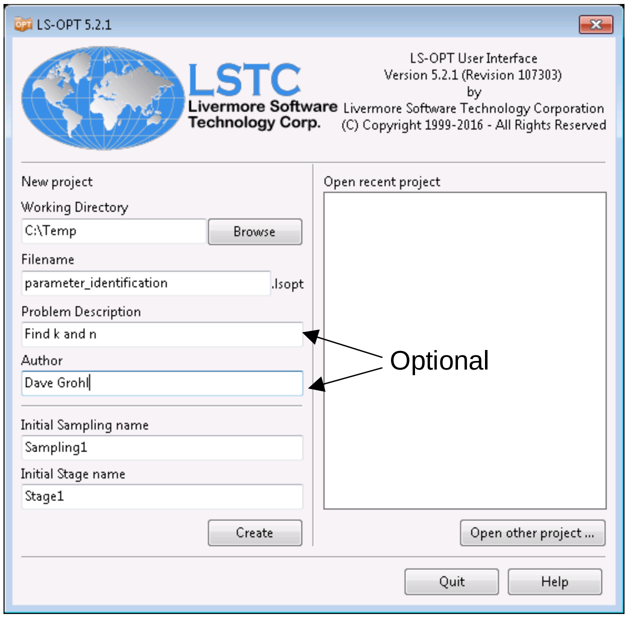
In this exercise, we want to extract the force and the displacement from the simulation. Click on **Model > Display,** double-click **Database**. Select **History_Node**. In the Entity Selection, select both node **135** and **458**. The displacement will be measured between these two nodes. The nodes are located at a distance from each other which corresponds to the displacement measurement in the experiment. **NODOUT** is activated in **DATABASE_ASCII_option** to gather data from these node displacements.

Select **Cross_Section_Set** to display the section where the force will be measured. **SECFORC** is activated in **DATABASE_ASCII_option** to obtain force data from this section. Close LS-PrePost.

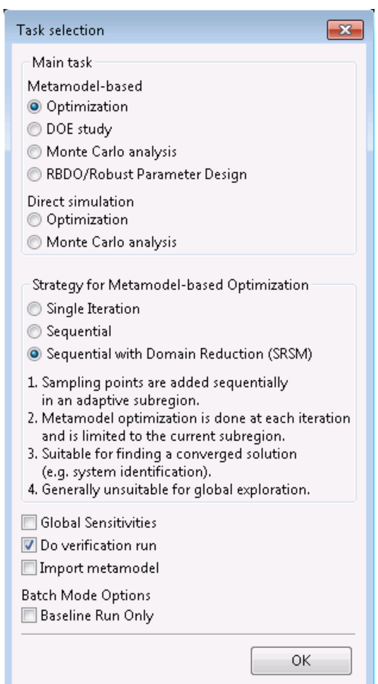


3 Setting up the parameter identification in LS-OPT

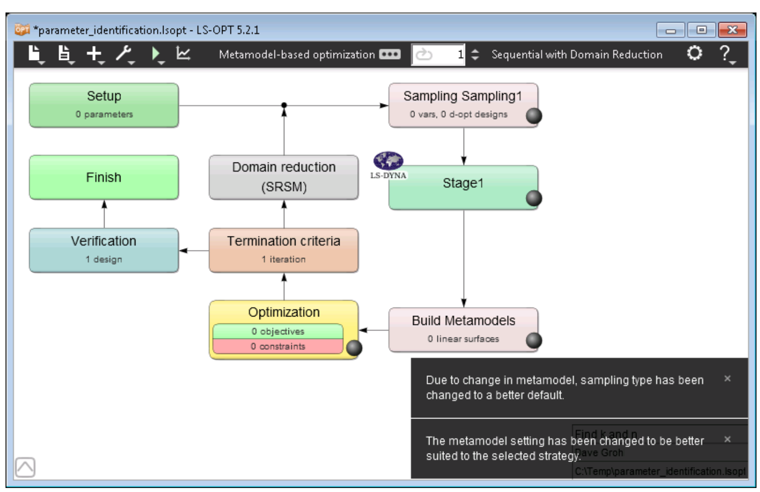
To set up the optimization do as follows.



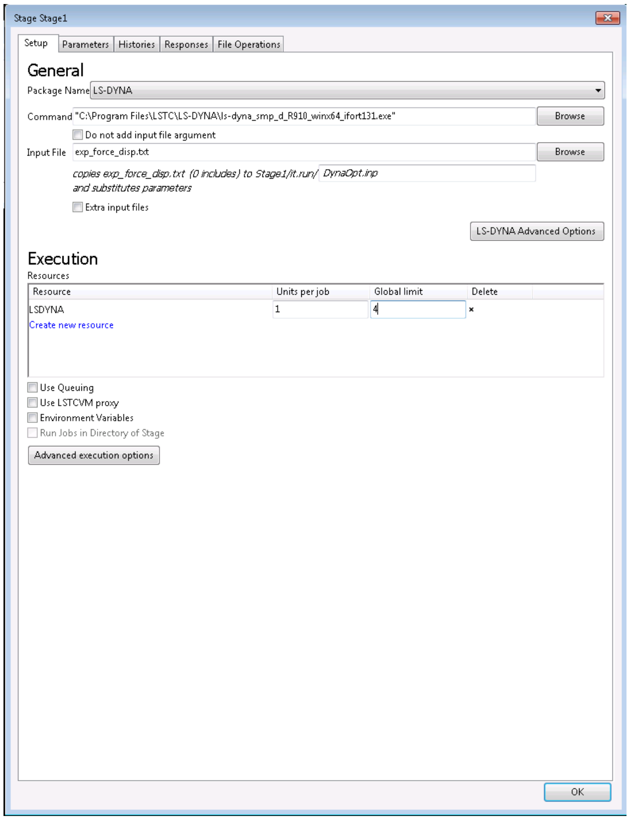
- Open **LS-OPT** by clicking from the start menu (in the LSTC folder). Either you can open an existing project or create a new one.
- Enter the path for your working directory and the name of the new file.
- Problem description and author are optional.
- Click **Create**.



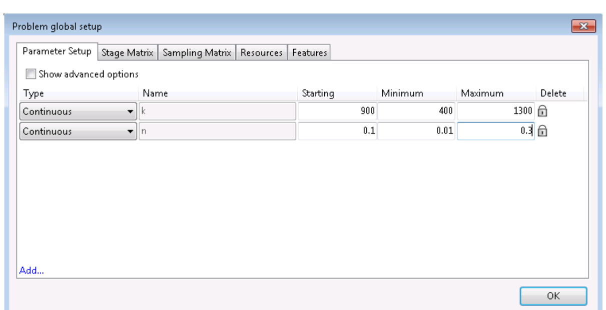
The next step is to choose the correct task for your study. Click on in the top menu. A parameter identification, is an optimization process where you are interested in a single optimal setting of your parameters. Therefore, the appropriate choice of task is the **Sequential Response Surface Method, SRSM**, change this and click **OK**. Notice how the flowchart changes slightly as you do this. You are also informed in the lower right corner about how your optimization settings have been given more appropriate choices for the selected task.



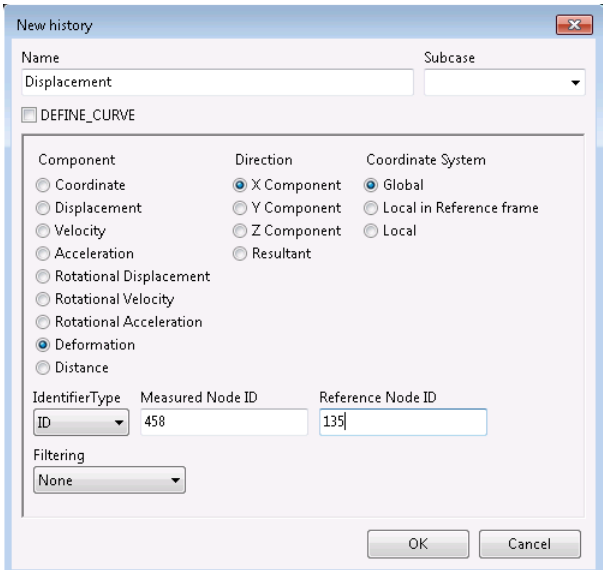
- Double-click on the **Stage1** box.
- Check that **Package Name** is **LS-DYNA** and change **Command** to the path for the LS-DYNA solver. Use double precision (since we are performing an implicit tensile test). Add quotes around the **Command** if it contains blank space character, as in the example below.
- Find your input file **tensile_test.k**.
- Change the **Global limit** to **4**, this implies that four simulations will be performed simultaneously.
- Click **OK**.



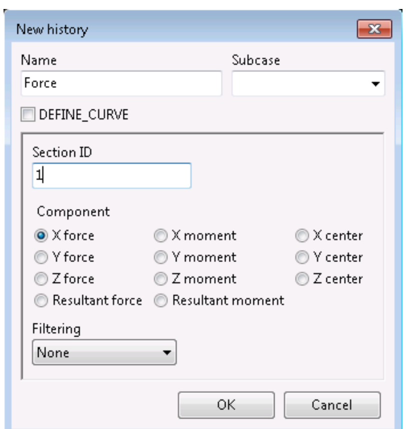
Note that two parameters automatically were identified by **LS-OPT** by double-clicking on the **Setup** box. Change the **Type** to **Continuous** and set **Starting**, **Minimum** and **Maximum** value as in the figure for the two parameters. When done, click **OK**.



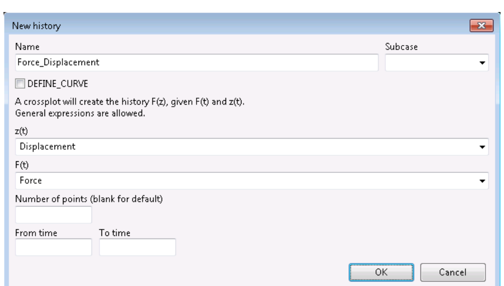
Apart from variables, we also need to specify responses that should be extracted from the simulations. In this case, we are interested in a force-displacement curve that we want to fit to the experimental force-displacement curve. Therefore, we have an output from the simulation which is the history of the displacement of two nodes.



- Double-click on the **Stage1** box again and select the **Histories** tab.
- Click on **NODOUT** and enter the values as in the figure.
- Define the displacement as the deformation between node **458** and **135** as in the figure to the left.
- Click **OK**.



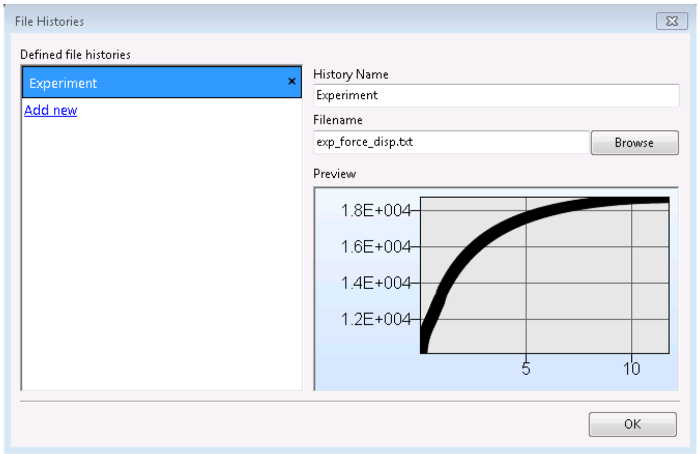
- The tensile force in the simulation is extracted by a cross-section set.
- Click on **SECFORC**, enter the values as in the figure.
- Click **OK**.

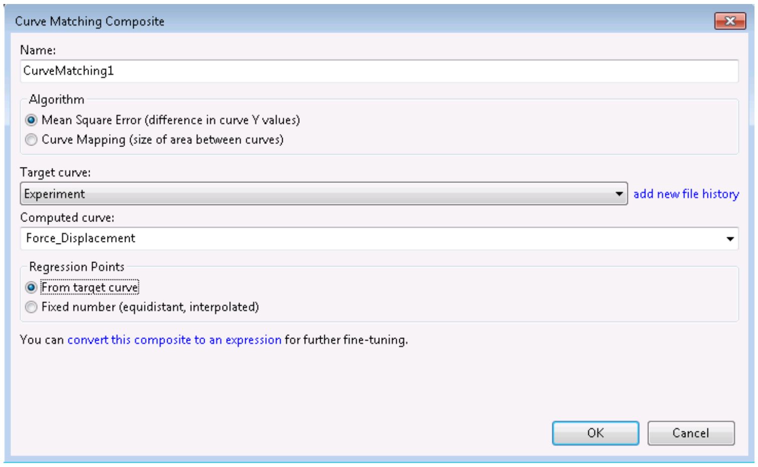


- When you have the two histories, force and displacement as functions of time, the thing left to do is to create the cross plot of the two to get the sought force-displacement relation.
- Click **Crossplot** (under Derived).
- Enter the values as in the figure (previously defined histories are available in pull down menus), click **OK**.
- Click **OK** in the Stage dialog.

The last step in the setup is to tell **LS-OPT** that the force-displacement curve from the simulation should be matched to the force-displacement curve from the experiment. This is achieved by adding a curve matching composite.

Click on the plus sign in the top menu and then **Add Composite**. Click **Curve Matching**. Change **Algorithm** to **Mean Square Error** (Curve Mapping could probably also work). Click **add new file history**, then **Add new**. Enter the name of the **History** and find the experimental result file **exp_force_disp.txt**. Click **OK**.





- Set the **Computed curve** to **Force_Displacement**.
- Click **OK** and then **OK** again.

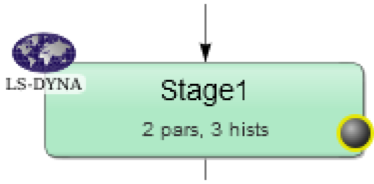
Finally, you must set the objective for the optimization:

- Click **objectives** in the **Optimization** box and select **CurveMatching1** as objective.
- Click **OK**.

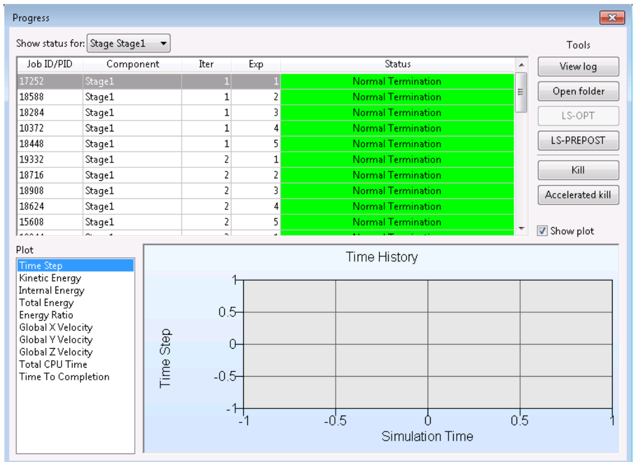
- Double-click on the **Termination criteria** box, change **Maximum number of iterations** to **10**. Keep the other criteria as the default values. If these termination criteria are reached first, LS-OPT will terminate and not perform the maximum number of iterations.
- Click **OK**.

4 Run the optimization in LS-OPT

Start the optimization by clicking on  in the top menu and then **Normal run**.



If you double click on the led belonging to **Stage1**, we will get a progress window. Here you can monitor the progress of all the simulations. By selecting one of the simulations, you can access the folder (**Open folder**) for that simulation, if you are interested in looking at its corresponding files.



You may also look at the log file (**View log**) for a simulation. The log file is basically the output from the simulation with some extra information related to the LS-OPT progress. This might be useful for debugging.

A third option is to open the selected simulation (**LS-PREPOST**) using LS-PrePost. When clicking on this button, you are also asked to select which file to open with LS-PrePost, whether it is the input file, the d3plot file or some other file.

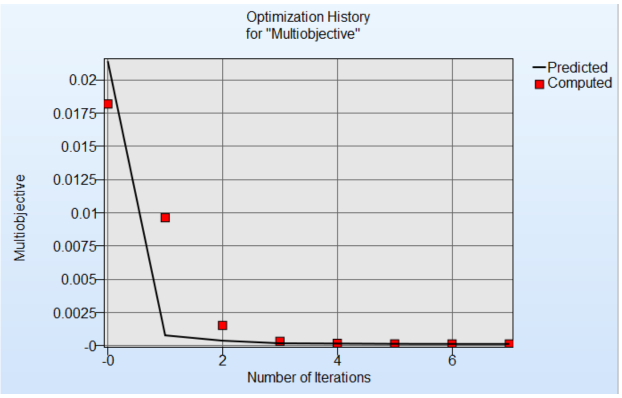
Five simulations (Exp in Progress window) are performed for every iteration. This is stated in the **Sampling** box and can also be changed.

The optimization might take some minutes. When this dialog appears, the optimization is done.

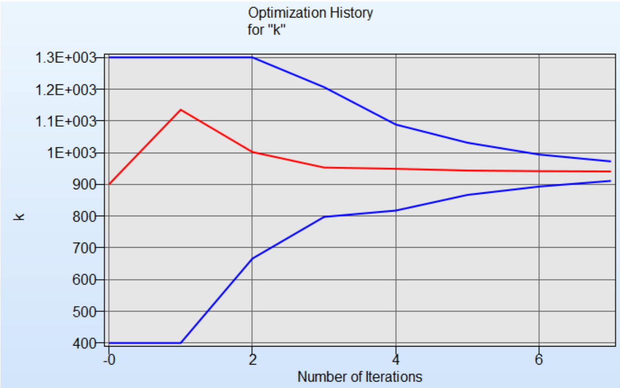
5 Analyze the results from LS-OPT

5.1 Convergence history

In this case, seven iterations were performed. After this, a final verification run was simulated to obtain the optimal solution. **Note** that the results from this tutorial might differ from your results. Close the **Progress** window. Look at the results from the parameter identification by opening the viewer in the top menu. There you have a number of different results to choose from.



- Start by clicking **History** under **Optimization**.
- First of all, you can see how the objective has changed i.e. how the mean square error has decreased.

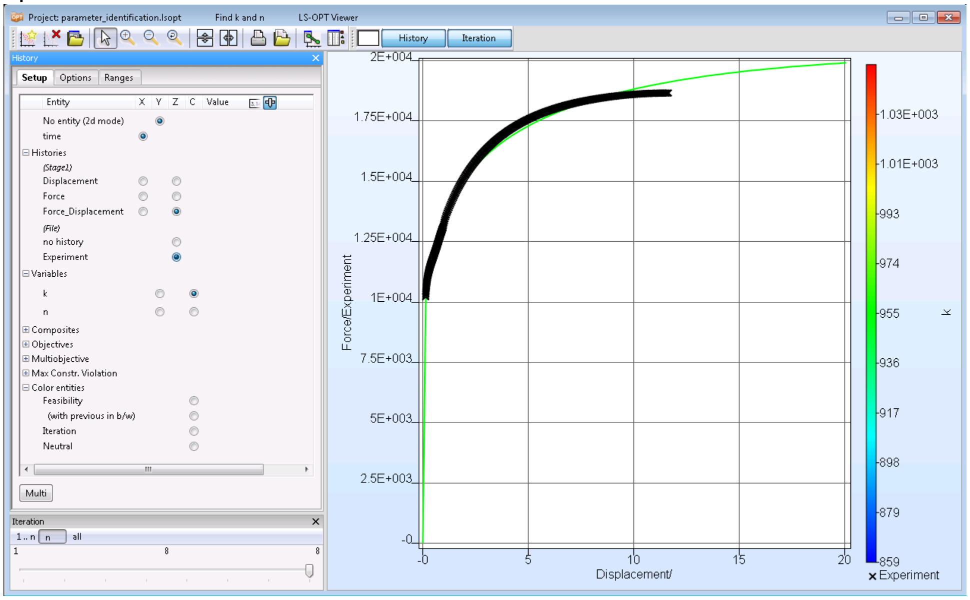


- You can also see how the values of the two variables K and n have changed. Click on variable k in the left column. The blue lines are the variable bounds and the red line is the actual value of k.
- Close the window.

5.2 Check the quality of the solution

To check the quality of the fitted curve/parameters by comparing the target curve with the curve for the fitted parameters k and n:

- Open **Histories** under **Simulations** in the viewer .
- Activate **Force_Displacement** and **Experiment** in the **Z** column. What you see here is the optimal solution which is obtained in the verification run after the final iteration.



By using the Iteration toolbar in the lower left corner, you can change the iteration step. By clicking on the curves, you can look at the variable values associated with that particular simulation, as well as response values. You can switch between different simulations in that iteration by clicking on the different simulations performed, or you can do a multiple select (plus sign in Point selection box) to be able to do a comparison between the simulations.

6 Summary and where to learn more

This tutorial has given you a short introduction on how to perform a parameter identification in LS-OPT.

For more examples, tutorials, videos, FAQs, HowTos etc. visit http://www.lsoptsupport.com/.