



# So.ph.i. - SOfware for Phenomenological Implementations - v. 4.5

## User's Guide

### Contents

Program characteristics .....	2
Installation .....	2
Usage .....	2
Angular module .....	2
Hold-down module .....	4
Screw module .....	5
Diagonal module .....	6
Users commands .....	7
Main menu commands .....	7
“Tools” section commands .....	8
“Control parameters” panel .....	9
Usage remarks .....	9
Elastic Cycles .....	9
Auto-calibration algorithm .....	10
Optimization algorithms .....	11
Genetic optimization .....	11
Scan optimization .....	12
Using Pivot option .....	13

## Program characteristics

So.ph.i. is a software that helps calibrating skeleton curves and hysteresis loops for numerical implementations. The software is designed for zero-length non-linear springs and it's specialized in wooden connectors (angulars, hold-downs and screws - single or in parallel - and diagonal springs for wooden walls).

This software is free to download and to use. Please send any comment or suggestions to [giovanni@rinaldin.org](mailto:giovanni@rinaldin.org).

The file extension used for saving data is the ".sph" format, or the compressed one ".sph.zip" (suggested). Sophi supports ZIP compression to save and open data files. Each ZIP archive contains three files:

1. a ".sph" file as mentioned before;
2. a ".sph.jpg" file that stores the image used for calibration;
3. a ".sph.inp" file that contains the input for the ABAQUS model in which subroutines are implemented. Please check the version of the routine used by Sophi in the menu ?/Info... before launching any analysis in ABAQUS.

When you open a ZIP archive in Sophi these three files remains extracted, in order to speed up the next openings. Only if you save the file again with the same name the original data files will be deleted. It is recommended to use a specific folder to store Sophi data.

## Installation

Requirements:

- on Windows Vista/7, please disable UAC (User Access Control) to make Sophi work properly;
- use dot "." as decimal separator;
- have .NET Framework 2.0 installed.

At the end of the installation wizard, a link to Sophi software will available in Start menu and on the user's desktop. Please check periodically updates with the command ?/Check updates...

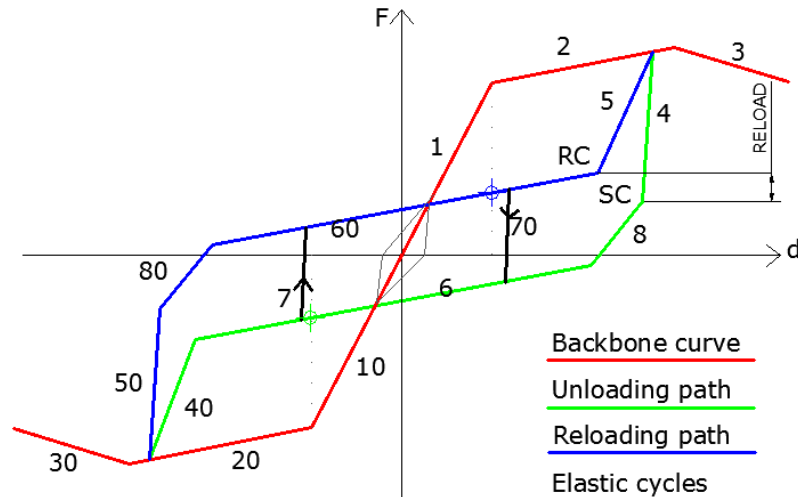
## Usage

First, choose a curve type form the menu. In the new windows, load the force - displacement curve desired with the button "Open image...". Then press "Plot data" to obtain the graphical representation of the curve, parameterized through the control parameters on the right. Finally, you can choose to save the file with the proper command. The "Load data" button is used to open previously saved calibrations.

After the previous commands, press Execute plot several hysteresis loops of the law considered. It is useful to see the effects of all parameters.

### **Angular module**

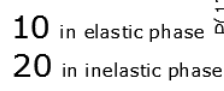
This module is suitable for modelling angular brackets; all parameters are relative to this polygonal model:



The input parameters required are:

1.  $K_{el}$ : Elastic stiffness;
2.  $F_y$ : Yielding force;
3.  $K_{1pl}$ : First inelastic stiffness (hardening branch);
4.  $F_{max}$ : Maximum force allowable;
5.  $K_{2pl}$ : Second inelastic stiffness (softening or hardening branch);
6.  $K_{sc}$  factor: it sets the unloading stiffness of branches 4 and 50 multiplying the elastic stiffness by this factor;
7. RC parameter: it sets the lower limit of branches 5 and 40 multiplying the force value before entering unloading path;
8. SC parameter: it sets the lower limit of branches 4 and 50 multiplying the force value before entering unloading path;
9.  $U_{ult}$ : ultimate displacement of the law; it contains always the positive one;
10. PinDeg: (Pinching degradation parameter) it controls the stiffness of pinching branches (6 and 60);
11. Kdeg: (Strength degradation parameter) it controls the linear degradation of unloading stiffness once entered in inelastic field;
12.  $\alpha$ : exponential strength degradation parameter based on dissipated energy;
13.  $\beta$ : exponential strength degradation parameter based on maximum displacement;
14.  $\gamma$ : linear strength degradation parameter;
15. 8<sup>th</sup> branch negative start force;
16. 8<sup>th</sup> branch stiffness coefficient, multiplies the elastic stiffness;
17. NoElCyc: if equal to 1, no elastic cycles are permitted in the analysis;
18.  $K_{5-rdc}$ : it sets the unloading stiffness of branches 5 and 40 multiplying the elastic stiffness by this factor.

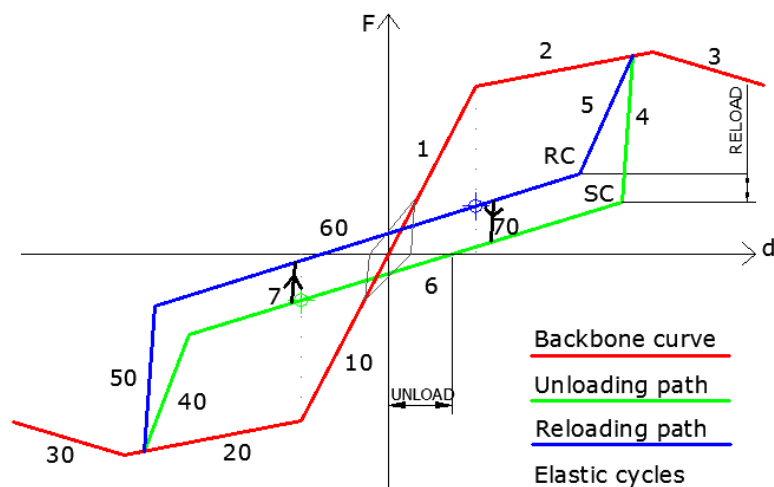
For the diagonal spring module the same parameters are needed.



- 4

## Screw module

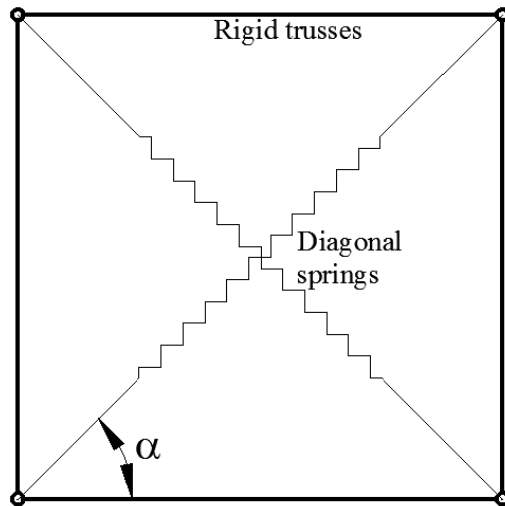
For screws modelling:



1.  $K_{el}$ : Elastic stiffness;
2.  $F_y$ : Yielding force;
3.  $K_{1pl}$ : First inelastic stiffness (hardening branch);
4.  $F_{max}$ : Maximum force allowable;
5.  $K_{2pl}$ : Second inelastic stiffness (softening or hardening branch);
6.  $K_{sc}$  factor: it sets the unloading stiffness of branches 4 and 50 multiplying the elastic stiffness by this factor;
7. RC parameter: it sets the lower limit of branches 5 and 40 multiplying the force value before entering unloading path;
8. SC parameter: it sets the lower limit of branches 4 and 50 multiplying the force value before entering unloading path;
9.  $U_{ult}$ : ultimate displacement of the law: it contains always the positive one;
10. PinDeg: (Pinching degradation parameter) it controls the stiffness of pinching branches (6 and 60);
11.  $k_{deg}$ : (Strength degradation parameter) it controls the linear degradation of unloading stiffness once entered in inelastic field;
12.  $\alpha$ : exponential strength degradation parameter based on dissipated energy;
13.  $\beta$ : exponential strength degradation parameter based on maximum displacement;
14.  $\gamma$ : linear strength degradation parameter;
15.  $K_{5-rdc}$ : it sets the unloading stiffness of branches 5 and 40 multiplying the elastic stiffness by this factor;
16. NoElCyc: if equal to 1, no elastic cycles are permitted in the analysis.

## **Diagonal module**

The diagonal module works for modelling of walls through diagonal springs in light-frame buildings. It uses the angle bracket parameters and gives as output the characteristics projected by the angle specified by the user by setting the parameters W (Width) and H (Height).



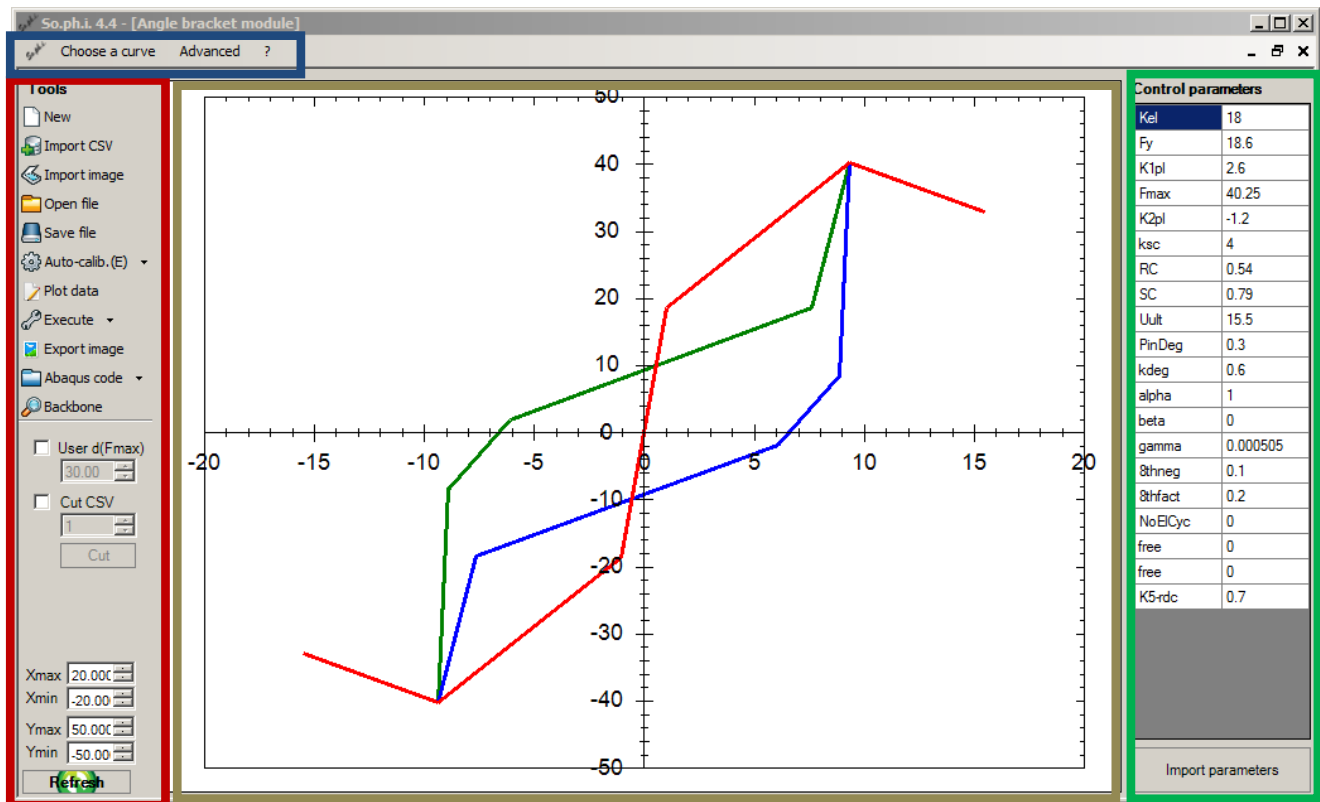
Force and stiffness are calculated as:

$$k_{diag} = \frac{k_{calibration}}{2 \cos^2(\alpha)}$$

$$F_{diag} = \frac{F_{calibration}}{2 \cos(\alpha)}$$

where  $\alpha$  is the angle between the diagonal spring and the horizontal truss.

## Users commands



Main menu

Tools

Control parameters

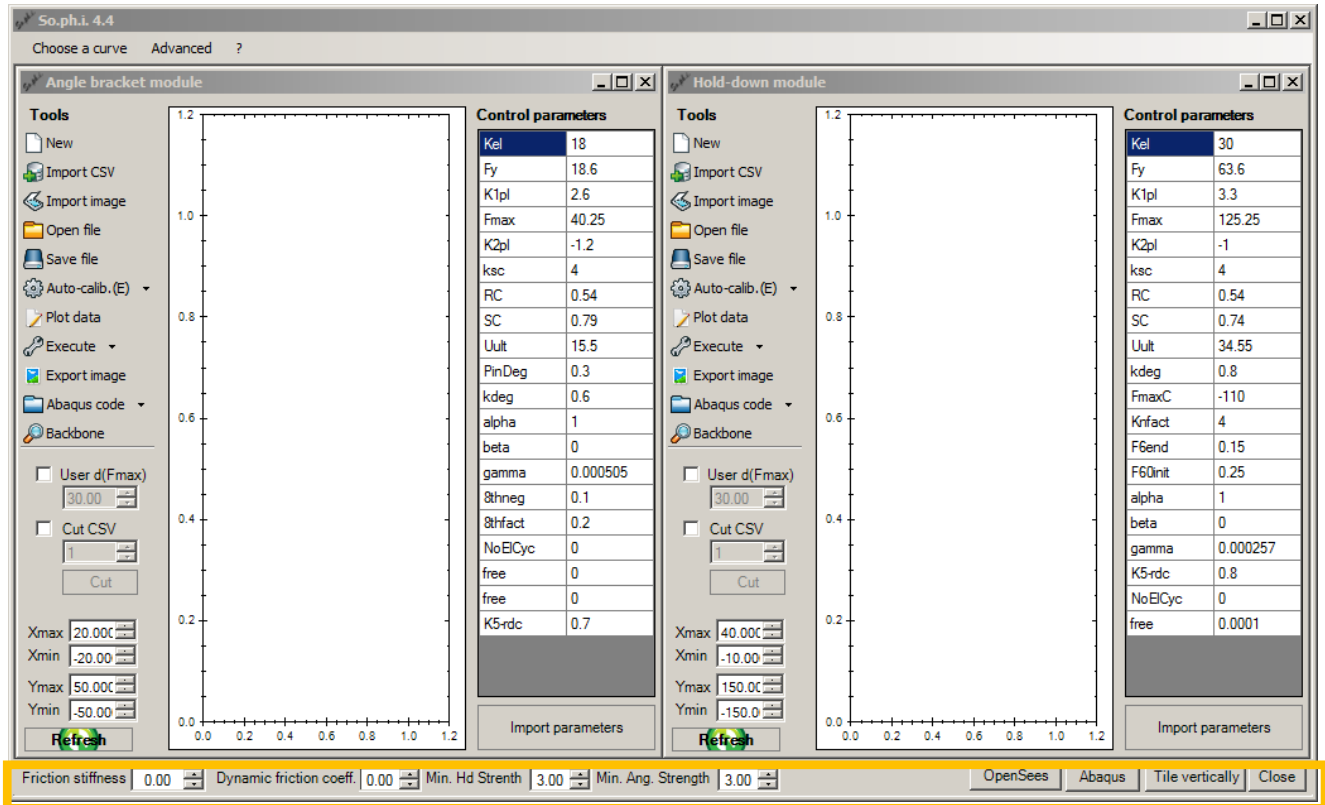
Viewport

### Main menu commands

*Choose a curve menu:* it is used to choose the desired module or to exit.

- Angle bracket:* display the module used to calibrate symmetric shear spring
- Screw:* display the module used to calibrate tension-contact spring
- Hold-down:* display the module used to calibrate tension-contact spring
- Diagonal spring:* display the module used to calibrate the diagonal springs
- Ang-Hd combo:* display the modules for the calibration of a composite spring (Angle bracket in horizontal shear and Hold-down in axial direction)
- Screws combo:* display the modules for the calibration of a composite spring (screw in horizontal shear and vertical shear) using two single Screw modules
- Exit:* quit the program.

When calling a “combo” modules set, an additional bar is displayed at the bottom of the viewport:



## Combo ribbon

The parameters in the “Combo ribbon” are:

- Friction stiffness*: it sets the initial stiffness for the friction model (the branch from zero force to friction force)
- Dynamic friction coeff.*: it sets the friction coefficient to be used during the analysis
- Min. Hd Strength*: it sets the minimum yielding resistance reachable when in the strength domain for the axial spring. Set it to the yielding strength to remove the domain feature.
- Min. Ang. Strength*: it sets the minimum yielding resistance reachable when in the strength domain for the shear spring. Set it to the yielding strength to remove the domain feature.
- OpenSees*: compile the input for the combo spring for OpenSees
- Abaqus*: compile the input for the combo spring for Abaqus
- Tile vertically*: tile the 2 windows vertically
- Close*: close the Combo ribbon.

## “Tools” section commands

**New** Initialise a new calibration session

**Open** Open a previously saved sph or sph.zip file.

**Save** Save the current calibration in the sph file you specify

**Import CSV** Import a numerical data series in 2 columns (displacement as first and force as second)

**Import image** Import an image of an experimental force-displacement plot, in which the borders have been previously cut to fit the margins



**Auto-calib. / EN method** Execute an automatic calibration considering the rules of the EN 12512:2001 for the backbone curve

**Auto-calib./Mod method** Execute an automatic calibration considering the rules of the EN 12512:2001, but imposing that the first plastic branch passes for the real maximum in strength

**Auto-calib./Degr. Params** Execute an optimization, following the ranges specified in the Advanced menu window, of the strength degradation params. See the dedicated chapter hereinafter.

**Plot data** Plot the backbone and the unloading (in blue) and the reloading (in green) path followed by the hysteresis law from and to the maximum strength level.

**Execute / Sinusoidal disp.** Execute the routine containing the hysteresis model with the parameters set in the "Control parameters" box, with a sinusoidal displacement time-history with increasing amplitude.

**Execute / Experimental disp.** Execute the routine containing the hysteresis model with the parameters set in the "Control parameters" box, with the real displacement history found in the CSV data series. This command produces a graph containing also the total energy difference between experimental and numerical behaviour.

**Execute / Export num. data** Export the numerical data obtained from the command "Execute / Experimental disp." in a CSV file specified by the user.

**Export image** Export the current viewport in a JPG or a BMP image.

**Abaqus code** Compile the spring characteristics in the Abaqus format and display them in a new window.

**OpenSees code** Compile the spring characteristics in the OpenSees format and display them in a new window.

**Backbone** Display the points of the calibrated backbone curve.

**Refresh** Refresh the current view with the upper and lower axes limits specified

### ***"Control parameters" panel***

It contains the set of parameters needed by the selected spring. The results of the calibration can be seen here; each value can be modified by the user.

## **Usage remarks**

The program exports all the springs characteristics ready to be used in ABAQUS subroutine version U1125, made for wooden connections. Such formatted values are in the file ***filename.sph.inp***.

Finally, to close the program is necessary to press the X button two times (the first one is needed to close all the module windows).

## ***Elastic Cycles***

Elastic cycles contributes to the total dissipated energy on a cyclic tests and they are designed to be used in single connector calibrations. In structures models it is suggested to deactivate them by imposing the parameter NoElCyc, present in every hysteresis law proposed, equal to 1. If not, Abaqus solver can give you error 693.

## Auto-calibration algorithm

The elastic and the first inelastic branch are determined in accordance with EN 12512:2001 directives. After, the elastic stiffness is corrected so the elastic branch can pass through origin. The second inelastic stiffness is calculated from the point of maximum force and ending to a point whose displacement the maximum absolute displacement (the Ultimate displacement) and the force associated with it is the Maximum force reduced by 20%.

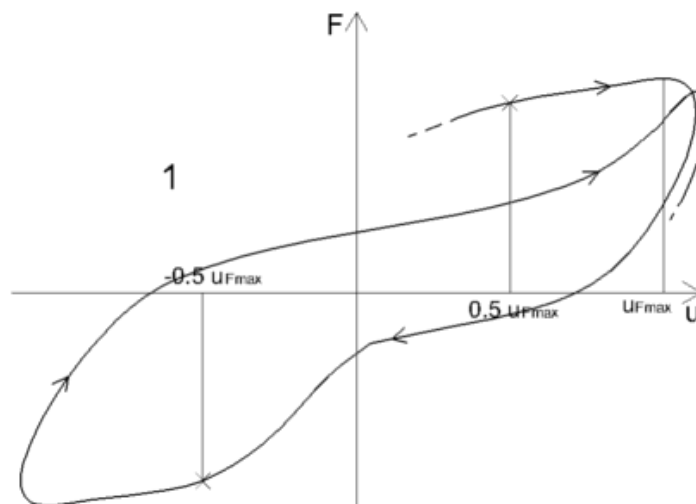
Sometimes EN 12512:2001 directives may lead to excessively high maximum forces. Therefore the possibility of a second choice has been inserted in the "Mod method". In this mode, the maximum force is assumed to be the real maximum found in loaded data. Hence, first inelastic stiffness is adjusted.

All other branches are detected using points that are believed to be (generally) revealing the stiffness of a branch:

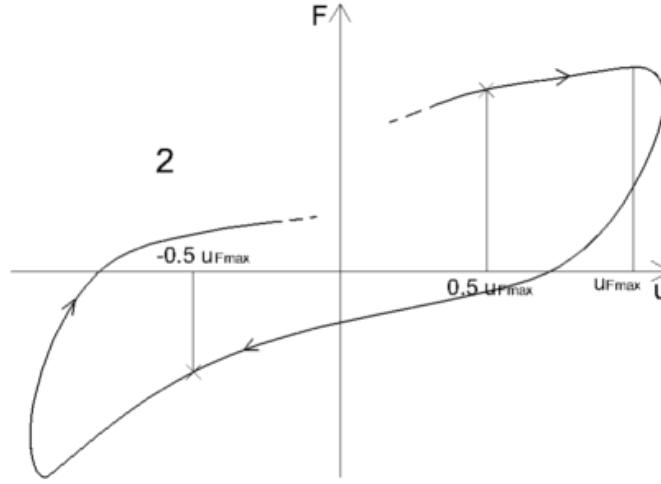
- unloading stiffness is calculated using points at 85% and 60% of the unloading branch nearest to the maximum strength;
- reloading stiffness is calculated using points at 90% and 70% of the of the reloading branch nearest to the maximum strength;
- branch #6 is found locating points at +40% and -40% of the displacement associated to the Maximum force (this is true for Angle-bracket and Screw modules, Hold- down indeed uses +10% and +50%);
- branch #8 is set tangent to loaded data and its stiffness is calculated as the geometric average of stiffnesses from branches #4 and #6.

All these parameters may be adjusted if the law doesn't have a shape similar to those shown in the figures above. This can happen if certain stiffnesses are negative or outside a tolerance interval.

So.ph.i. distinguishes between a backbone branch and a reloading branch comparing two forces as shown in the next two figures:



**First case:** the two absolute values are almost the same. That means loaded data is drawing a backbone curve with negative forces. Sophi will move forward another half-cycle to find a reloading series.



**Second case:** the two absolute values are quite different. That means loaded data draws a reloading branch in the half-cycle immediately following the maximum force.

Once calculations has been done, the new piecewise linear law will be displayed automatically on the table on the right side of the module's window.

## Optimization algorithms

The optimization algorithms included in So.ph.i. are only available with data series loaded.

### Genetic optimization

The optimization algorithm included in So.ph.i. is used to minimise simultaneously:

- the difference of the force level, point by point, between the experimental and the numerical curve;
- the difference in total energy, calculated for both curves and for every point  $i$  as follow:

$$E_{tot,i} = E_{tot,i-1} + \frac{(F_i + F_{i-1})(u_i + u_{i-1})}{2}$$

with  $F$  as force and  $u$  as displacement.

In order to do this, a genetic algorithm developed at University of Trieste has been included in the software. This algorithm tries to minimise the function  $f$ , running repeatedly the numerical simulation and varying the parameters  $\alpha, \beta, \gamma$  of the spring as specified by the user.

$$f = \frac{\sum_i (F_{num,i} - F_{exp,i})^2}{n \cdot F_{exp,max}^2} + \left( \frac{E_{tot,num} - E_{tot,exp}}{E_{tot,exp}} \right)^2$$

where:

$F_{num,i}$  is the force of  $i$ -th point in the numerical curve

$F_{exp,i}$  is the force of  $i$ -th point in the experimental curve

$F_{max,exp}$  is the maximum experimental force

$E_{tot,num}$  is the total energy for the numerical curve

$E_{tot,exp}$  is the total energy for the experimental curve

$n$  is the total number of points in the curve.

For further information about the optimizer, please contact the developer C. Chisari (corrado.chisariATgmail.com).

## Scan optimization

The scan optimizer seek for an optimum parameters set by varying each parameter in a certain range, specified by the user. The optimization algorithm in this case tries to minimize the function:

$$g_1 = \sqrt{\frac{\sum_i (F_{num,i} - F_{exp,i})^2}{n}}$$

$$g_2 = \frac{E_{tot,num} - E_{tot,exp}}{E_{tot,exp}} \cdot 100$$

where:

$F_{num,i}$  is the force of i-th point in the numerical curve

$F_{exp,i}$  is the force of i-th point in the experimental curve

$n$  is the total number of points in the curve

$E_{tot,num}$  is the total energy for the numerical curve

$E_{tot,exp}$  is the total energy for the experimental curve.

The user must specify first the range in which each parameter will be varied in the following window, fulfilling the columns Min, Max and Step for the desired rows.

Name	Value	Consider	Min	Max	Step
Kel	18	exclude			
Fy	18.6	exclude			
K1pl	2.6	exclude			
Fmax	40.25	exclude			
K2pl	-1.2	exclude			
ksc	4	independent	3	5	0.1
RC	0.54	exclude			
SC	0.79	exclude			
Uult	15.5	exclude			
PinD...	0.3	exclude			
kdeg	0.6	exclude			
alpha	1	exclude			
beta	0	exclude			
gam...	0.00...	exclude			

The desired rows are selected changing the value “exclude” in the third column to:

- “dependent” if the parameter is affected, in its variation, by the variation of all the other active parameters
- “independent” if the parameter is NOT affected, in its variation, by any of the others.

Once OK button is pressed, the analysis starts and at the end the user is asked to accept or not the parameters that minimise the function  $g$ . This algorithm is much faster than the genetic one, and the user can select which function must be used.

Minimize

☐ Distance between curves

☒ Total energy (best for Pivot)

## Using Pivot option

The option “Pivot model” introduced since version 4.5 of So.ph.i. is useful to calibrate the springs in the general FEM solver SAP2000 (CSI Berkeley). This option does not permit to save the parameters used and activates a completely different dataset, presented in the following figure on the left.

KelP	18
FyP	18.6
Kp1P	2.6
FmaxP	40.25
Kp2P	-1.2
KelN	18
FyN	-18.6
Kp1N	2.6
FmaxN	-40.25
Kp2N	-1.2
UuP	15.5
UuN	-15.5
alpha1	100
beta1	0.2
alpha2	100
beta2	0.2

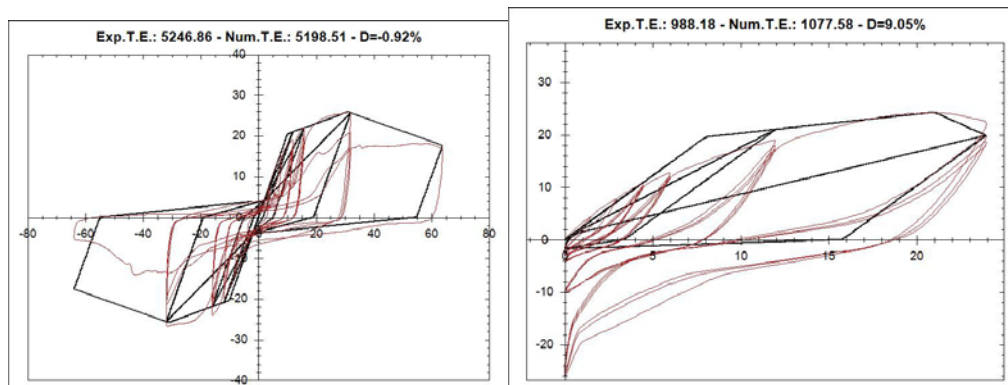
  

Backbone - Angle bracket module	
Displacement	Force
-15.5000	-32.8823
-9.3603	-40.2500
-1.0333	-18.6000
0.0000	0.0000
1.0333	18.6000
9.3603	40.2500
15.5000	32.8823

Show all points      Close

To use the parameters found in So.ph.i. in SAP2000 spring definition (Define/Link-Support properties...) the user has to copy the values contained in the “Backbone” window, and the alpha1, beta1, alpha2 and beta2 from the parameters list.

Differently from what SAP2000 does, the routine included in So.ph.i. has the definition of the ultimate displacement in the Pivot law, allowing some considerations in the total energy.



Finally, the routine included in So.ph.i. for the Pivot rule is NOT the same than the one in SAP2000, the uniformity of results is not guaranteed in every case.