Giovanni Rinaldin 2011 - 2017

So.ph.i. - SOftware for Phenomenological Implementations – for v. 4.6 and above

User's Guide

Contents

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 .
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.

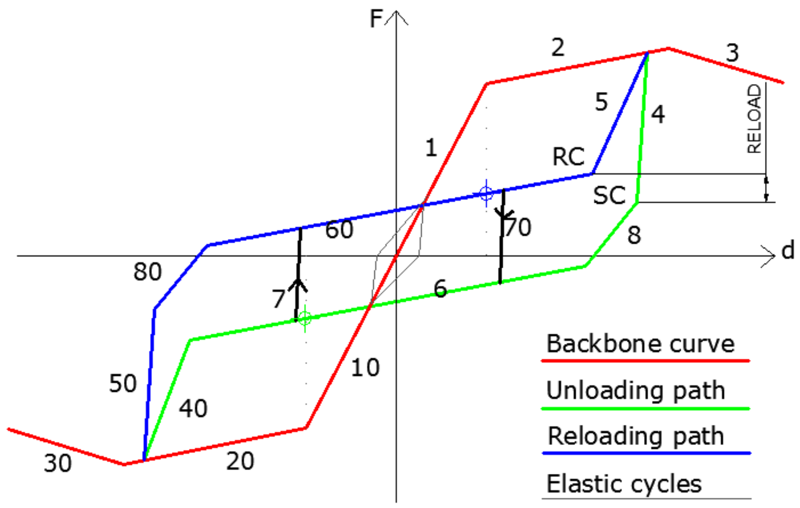
At the end of the installation wizard, a link to Sophi software will available in Start menu and on the user's desktop. Please check periodically updates with the command ?/Check updates…

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:



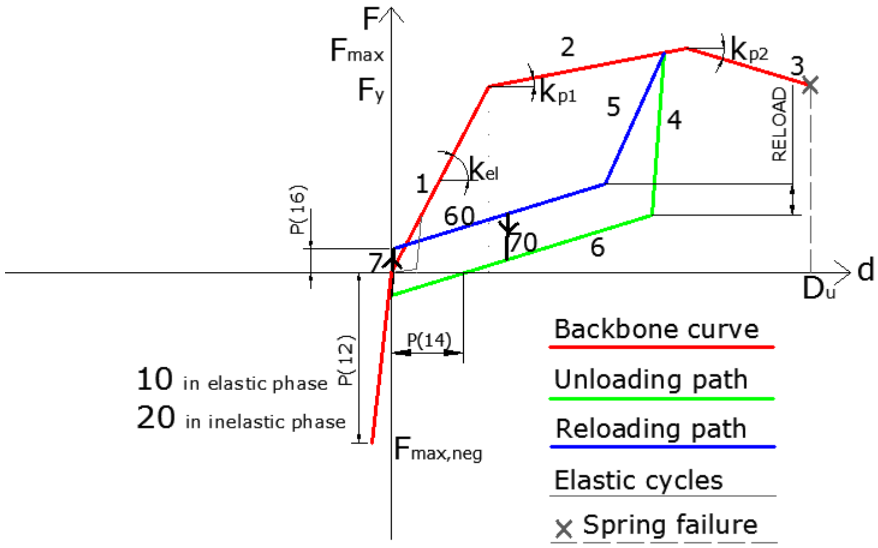
The input parameters required are:

1. Kel: Elastic stiffness;
2. Fy: Yielding force;
3. K1pl: First inelastic stiffness (hardening branch);
4. Fmax: Maximum force allowable;
5. K2pl: Second inelastic stiffness (softening or hardening branch);
6. Ksc 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. Uult: 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^{th} branch negative start force;
16. 8^{th} branch stiffness coefficient, multiplies the elastic stiffness;
17. NoElCyc: if equal to 1, no elastic cycles are permitted in the analysis;
18. K5-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.

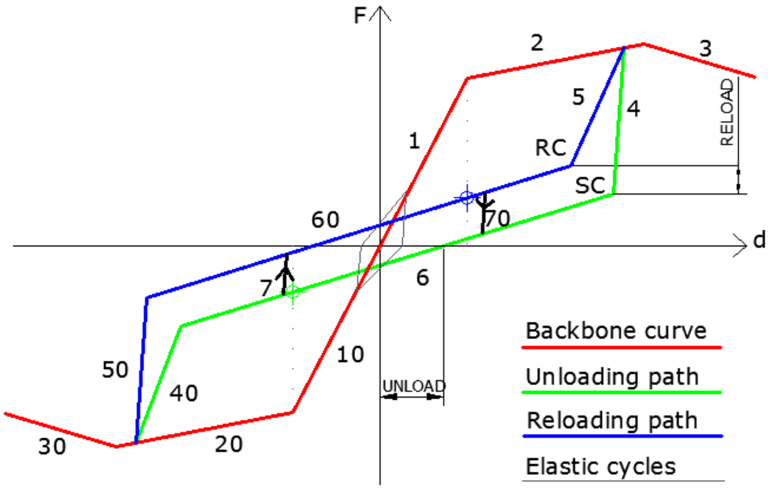
Hold-down module

For the hold-down module the parameters are:



1. Kel: Elastic stiffness;
2. Fy: Yielding force;
3. K1pl: First inelastic stiffness (hardening branch);
4. Fmax: Maximum force allowable;
5. K2pl: Second inelastic stiffness (softening or hardening branch);
6. Ksc 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. Uult: ultimate displacement of the law; it contains always the positive one;
10. Kdeg: (Strength degradation parameter) it controls the linear degradation of unloading stiffness once entered in inelastic field;
11. FmaxC: Maximum force in compression, always negative. It must be a reasonably high value, it models also the wood crushing;
12. Knfact: Stiffness coefficient for compression branch, multiplies the elastic stiffness;
13. F6end: Force value for ending of 6th branch;
14. F60init: Force value for starting of 60th branch;
15. alpha: exponential strength degradation parameter based on dissipated energy;
16. beta: exponential strength degradation parameter based on maximum displacement;
17. gamma: linear strength degradation parameter;
18. NoElCyc: if equal to 1, no elastic cycles are permitted in the analysis;
19. K5-rdc: 5th branch stiffness coefficient, multiplies elastic stiffness.

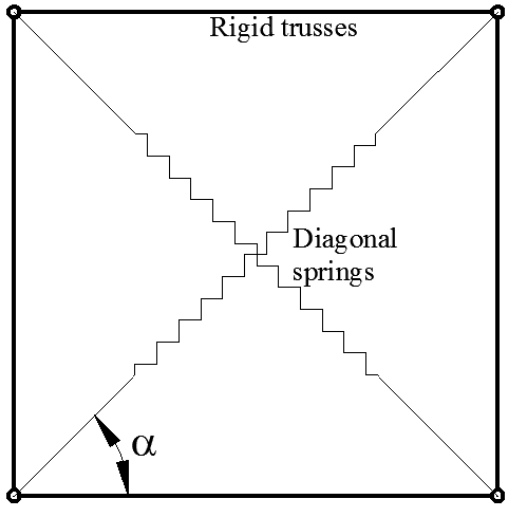
For screws modelling:



1. Kel: Elastic stiffness;
2. Fy: Yielding force;
3. K1pl: First inelastic stiffness (hardening branch);
4. Fmax: Maximum force allowable;
5. K2pl: Second inelastic stiffness (softening or hardening branch);
6. Ksc 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. Uult: 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. K5-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. Is 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).



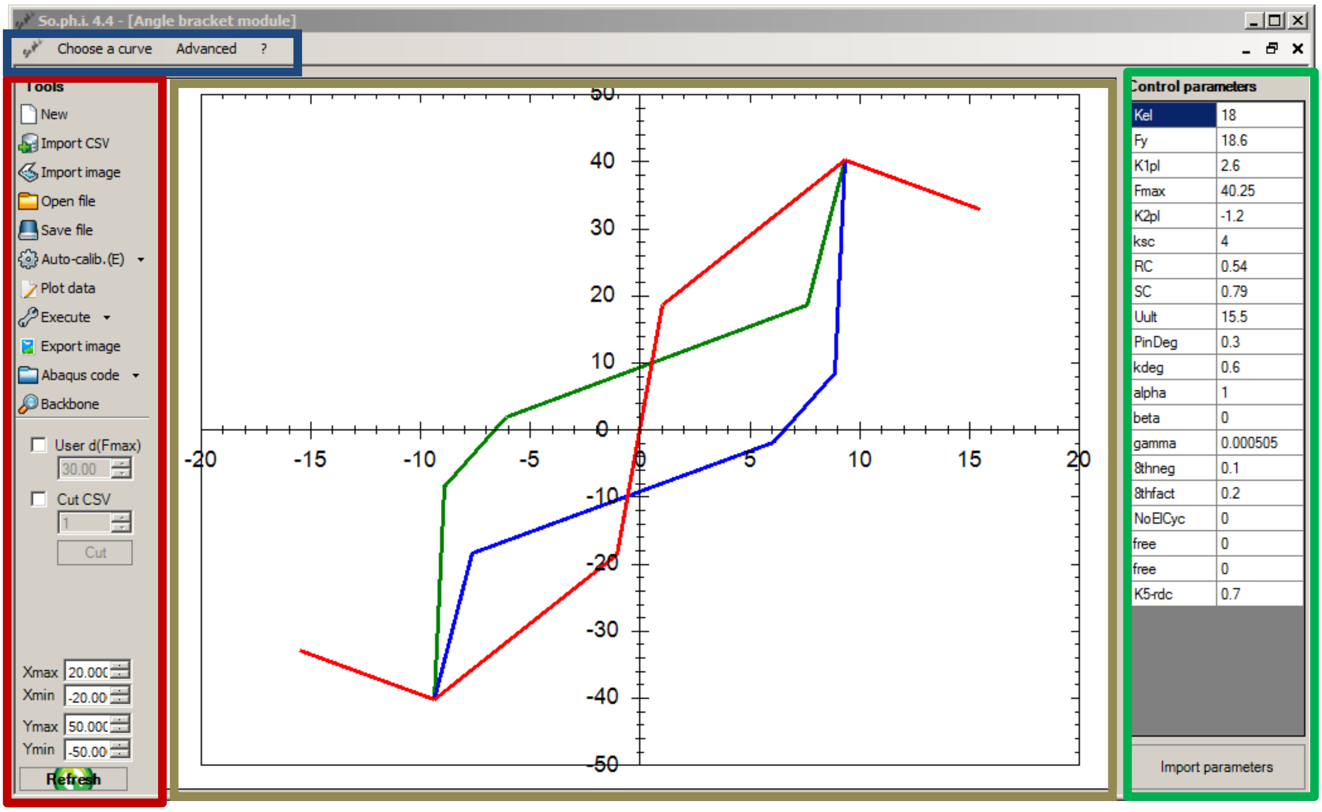
Force and stiffness are calculated as:

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

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

where α is the angle between the diagonal spring and the horizontal truss.

Users commands



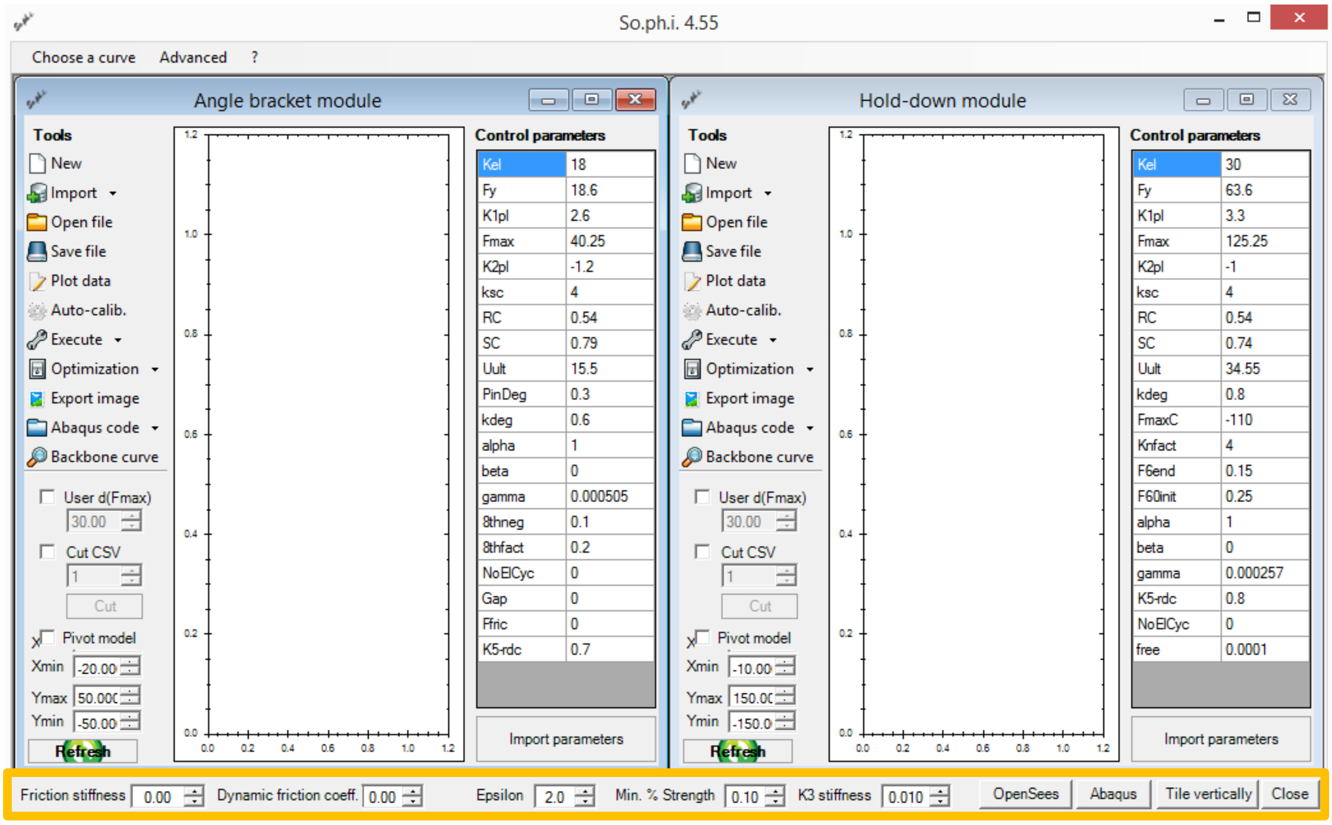
Main menu **Tools** **Control parameters** **Viewport**

Main menu commands

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

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

a. *Friction stiffness*: it sets the initial stiffness for the friction model (the branch from zero force to friction force)
b. *Dynamic friction coeff.*: it sets the friction coefficient to be used during the analysis
c. *Epsilon*: exponential value in the relationship for the strength domain. Set it to the 0 to remove the domain feature.
d. *Min. % Strength*: it sets the residual resistance one the connector has reached the failure.
e. *OpenSees*: compile the input for the combo spring for OpenSees
f. *Abaqus*: compile the input for the combo spring for Abaqus
g. Tile vertically: tile the 2 windows vertically
h. 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%.

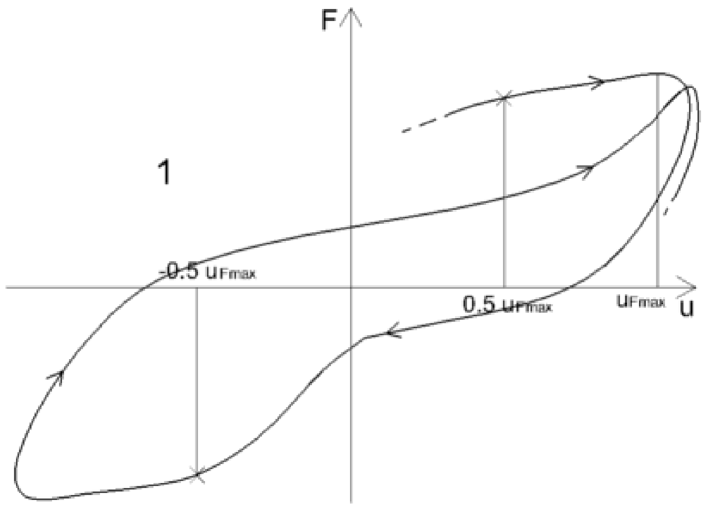
Sometimes EN 12512:2001 directives may lead to excessively high maximum forces. Therefore the possibility of a second choice has been inserted in the "Mod method". In this mode, the maximum force is assumed to be the real maximum found in loaded data. Hence, first inelastic stiffness is adjusted.

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

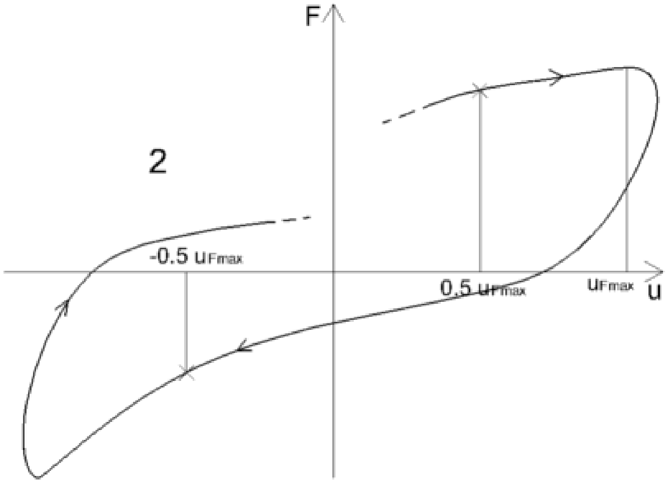
a. unloading stiffness is calculated using points at 85% and 60% of the unloading branch nearest to the maximum strength;
b. reloading stiffness is calculated using points at 90% and 70% of the of the reloading branch nearest to the maximum strength;
c. branch #6 is found locating points at +40% and -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%);
d. 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:

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

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

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 α, β, γ of the spring as specified by the user.

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

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:

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

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

where:

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

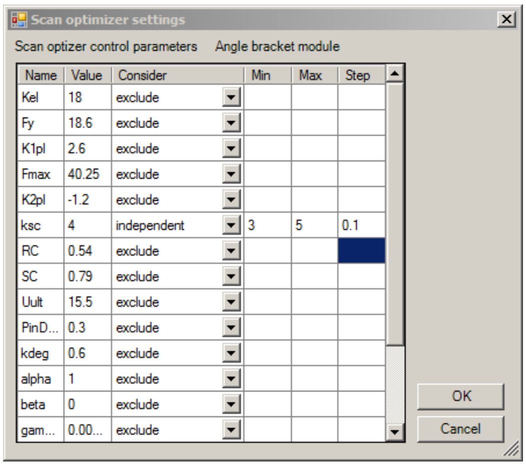
$F_{\exp,i}$ is the force of i-th point in the experimental curve

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.

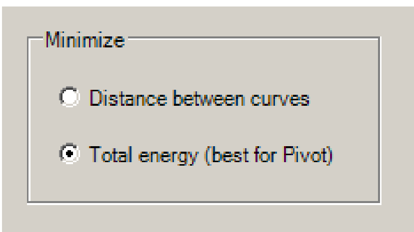
The user must specify first the range in which each parameter will be varied in the following window, fulfilling the columns Min, Max and Step for the desired rows.



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

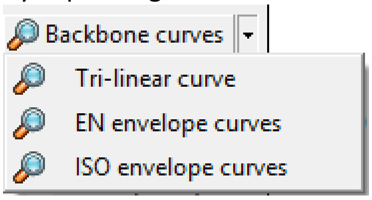
- "dependent" if the parameter is affected, in its variation, by the variation of all the other active parameters
- "independent" if the parameter is NOT affected, in its variation, by any of the others.

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 that the genetic one, and the user can select which function must be used.



ISO and EN backbones

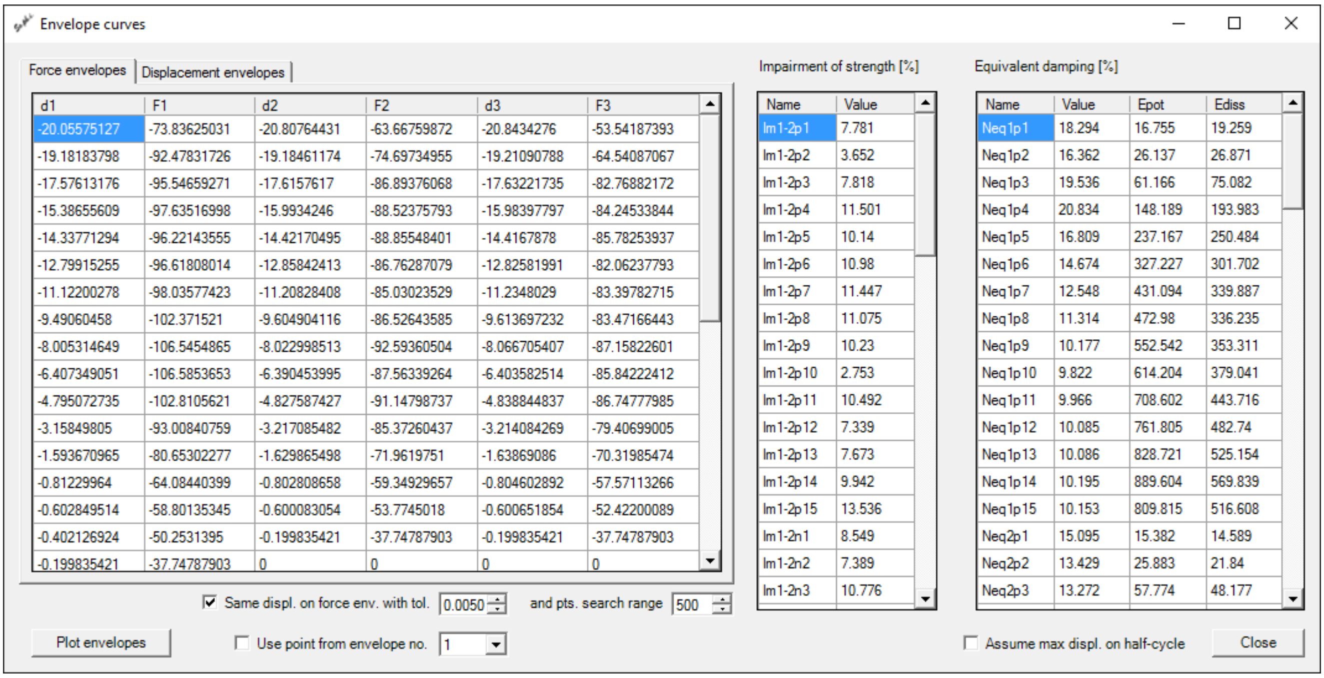
In the post-processing of an experimental test of a steel connector, the user is usually interested in having the backbone curves corresponding to the first 3 envelopes, calculated collecting the points at the same imposed displacement in a cyclic test.
By expanding the button "Backbone curves" the user can:



- obtain the tri-linear backbone (1st envelope) curve by (displacement; force) points with the command **Tri-linear curve**
- obtain the envelope curves by the code EN 12512:2006 (use ONLY if the test was performed with this protocol) by selecting **EN envelope curves**
- obtain the envelope curves by the code ISO 16670:2003 (use ONLY if the test was performed with this protocol) by selecting **ISO envelope curves**

A new windows will open, containing the following data:

1. 3 force envelopes (displ.;force) points
2. 3 displacement envelopes (displ.;force) points, obtained considering the maximum displacement per cycle;
3. The impairment of strength for each half loop considered (*Imp12p1* means the percentage impairment between the first points in the 1st and the 2nd positive envelopes)
4. The equivalent damping for each half loop considered (Neq1p1 represents the percentage equivalent damping corresponding at the 1st point in the 1st positive envelope curve, *Neq1p2* for the 2nd point in the 1st envelope curve and so on).



d1	F1	d2	F2	d3	F3
-20.05575127	-73.83625031	-20.80764431	-63.66759872	-20.8434276	-53.54187393
-19.18183798	-92.47831726	-19.18461174	-74.69734955	-19.21090788	-64.54087067
-17.57613176	-95.54659271	-17.6157617	-86.89376068	-17.63221735	-82.76882172
-15.38655609	-97.63516998	-15.9934246	-88.52375793	-15.98397797	-84.24533844
-14.33771294	-96.22143555	-14.42170495	-88.85548401	-14.4167878	-85.78253937
-12.79915255	-96.61808014	-12.85842413	-86.76287079	-12.82581991	-82.06237793
-11.12200278	-98.03577423	-11.20828408	-85.03023529	-11.2348029	-83.39782715
-9.49060458	-102.371521	-9.604904116	-86.52643585	-9.613697232	-83.47166443
-8.005314649	-106.5454865	-8.022998513	-92.59360504	-8.066705407	-87.15822601
-6.407349051	-106.5853653	-6.390453995	-87.56339264	-6.403582514	-85.84222412
-4.795072735	-102.8105621	-4.827587427	-91.14798737	-4.838844837	-86.74777985
-3.15849805	-93.00840759	-3.217085482	-85.37260437	-3.214084269	-79.40699005
-1.593670965	-80.65302277	-1.629865498	-71.9619751	-1.63869086	-70.31985474
-0.81229964	-64.08440399	-0.802808658	-59.34929657	-0.804602892	-57.57113266
-0.602849514	-58.80135345	-0.600083054	-53.7745018	-0.600651854	-52.42200089
-0.402126924	-50.2531395	-0.199835421	-37.74787903	-0.199835421	-37.74787903
-0.199835421	-37.74787903	0	0	0	0

Name	Value
Im1-2p1	7.781
Im1-2p2	3.652
Im1-2p3	7.818
Im1-2p4	11.501
Im1-2p5	10.14
Im1-2p6	10.98
Im1-2p7	11.447
Im1-2p8	11.075
Im1-2p9	10.23
Im1-2p10	2.753
Im1-2p11	10.492
Im1-2p12	7.339
Im1-2p13	7.673
Im1-2p14	9.942
Im1-2p15	13.536
Im1-2n1	8.549
Im1-2n2	7.389
Im1-2n3	10.776

Name	Value	Epot	Ediss
Neq1p1	18.294	16.755	19.259
Neq1p2	16.362	26.137	26.871
Neq1p3	19.536	61.166	75.082
Neq1p4	20.834	148.189	193.983
Neq1p5	16.809	237.167	250.484
Neq1p6	14.674	327.227	301.702
Neq1p7	12.548	431.094	339.887
Neq1p8	11.314	472.98	336.235
Neq1p9	10.177	552.542	353.311
Neq1p10	9.822	614.204	379.041
Neq1p11	9.966	708.602	443.716
Neq1p12	10.085	761.805	482.74
Neq1p13	10.086	828.721	525.154
Neq1p14	10.195	889.604	569.839
Neq1p15	10.153	809.815	516.608
Neq2p1	15.095	15.382	14.589
Neq2p2	13.429	25.883	21.84
Neq2p3	13.272	57.774	48.177

Finally, all the data can be copied to your spreadsheet by right-clicking each table and selecting **Copy table**.

The option "**Assume max displ. on half-cycle**" allows to calculate the envelopes on the maximum displacement on the semi-cycle considered. It can be used at user's discretion when the standard force envelopes are not satisfactory or when the experimental data is not clear enough.

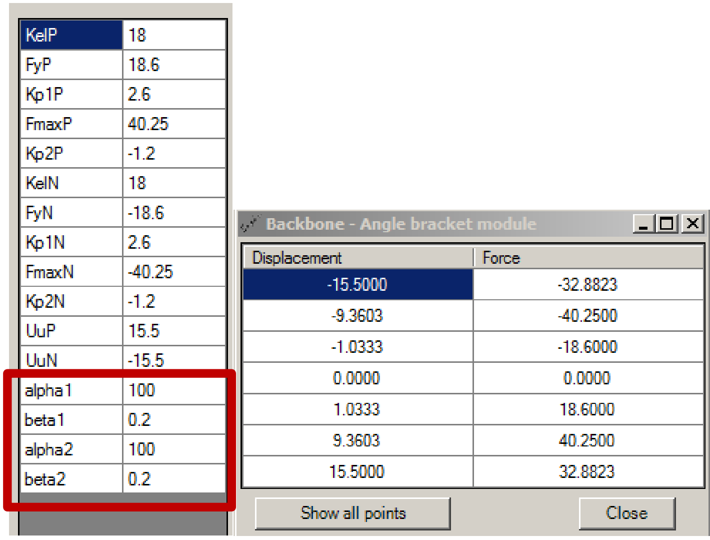
The option "**Same displ. on force env. with tol. 0.005 and pts. search range 500**" can be used to force the

displacement in each envelope to be equal. The minimum displacement for each triplet is assumed. The value 0.005 represents the searching tolerance on displacement in the range +/-500 of the experimental force/displacement point. This option can be used at user's discretion when the standard force envelopes are not satisfactory or when the experimental data is not clear enough.
When this last option is enabled, the additional option "**Use point from envelope no. 1**" allows to assume the reference displacement for each triplet on the chosen envelope.

IMPORTANT: when using such options, always press "**Plot envelopes**" to update results!

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

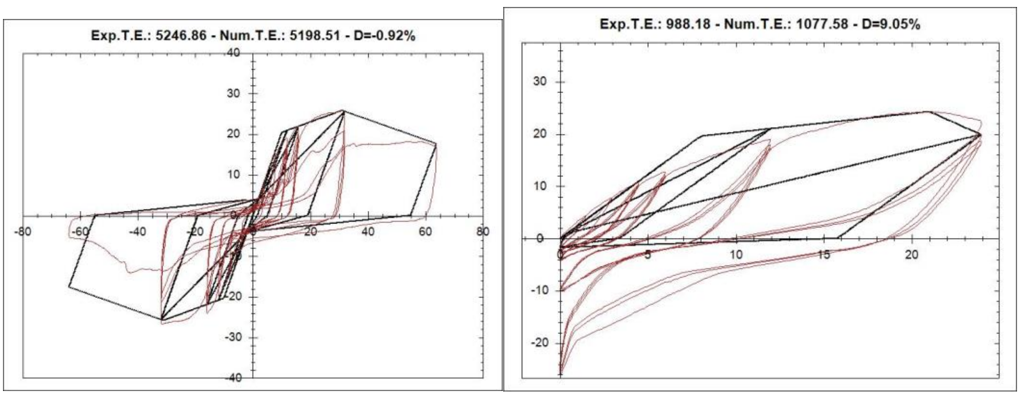
Backbone - Angle bracket module

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.

User's license

Rinaldin G., Amadio C., Fragiacomo M. (2013) A Component approach for the hysteretic behaviour of connections in cross-laminated wooden structures, Earthquake Engineering and Structural Dynamics, Wiley Online Library, DOI: 10.1002/eqe.2310