

September 2019

Version 0.7.x

Table of contents

Table of contents	2
Modelling	3
Assigning properties	4
Define tanks	5
Define external pressures	6
Load combinations	7
Optimization	8
Optimization iteration by predefined stiffeners	8
Single optimization	9
Multiple optimization	11
Span optimization	11
Export to JS	16

Modelling

Modelling is done in upper left corner.

Right click: select point

You can copy or move the selected point by shortcut or clicking Buttons.

Left click: select line

A line is made by right clicking two points (or input point number)

Input point coordinates [mm]		Add point (coords)
Point x (horizontal) [mm]:	<input type="text" value="0.0"/>	Copy point (relative)
Point y (vertical) [mm]:	<input type="text" value="0.0"/>	Move point (relative)

Input line from "point number" to "point number"		
From point number:	<input type="text" value="0"/>	Add line
To point number:	<input type="text" value="0"/>	

Delete lines and points (input line or point number)			
<input type="text" value="0"/>	Delete line	<input type="text" value="0"/>	Delete point

Speed up your modelling significantly by using the shortcuts:

CTRL-Z Undo modelling

CTRL-C Copy a selected point

CTRL-M Move a selected point

CTRL-Q New line between two selected points

CTRL-S Assign properties to a selected line

Assigning properties

Input properties manually or click the button indicated below to set the values. Values are set by clicking “Add structure to line”. This also applies to fatigue properties.

All beam sections are recorded. If you want to apply an existing, choose it from the drop down menu. Then press “Save and return structure”.

The screenshot displays the 'Define structure properties' window, which is divided into several sections for inputting structural data. Red callout boxes with arrows point to specific features:

- Define plate and stiffener properties.** Points to the 'Add line' button and the 'Delete lines and points' section.
- Define buckling calculation properties.** Points to the 'Find compartments' and 'Display current compartments' sections.
- Define fatigue properties.** Points to the 'Material yield [MPa]' and 'Internal, pressure from comp.' fields.

The main input area includes a table for structural and calculation properties:

span	pl_thk	web_h	web_thk	fl_w	fl_thk
700.0	18.0	400.0	12.0	250.0	14.0
[m]	[mm]	[mm]	[mm]	[mm]	[mm]

Below this table, there are fields for 'sig_y1', 'sig_y2', 'sig_x', 'tau_y1', 'stf type', and 'pressure side'. A 'Material yield [MPa]' field is set to 355.0. A 'Select structure type ->' dropdown menu is also present.

The right side of the window features a 'Define structure properties here --' section with a 'type:' dropdown set to 'T'. Below this, dimensions for 'Plate thk.', 'Web height', 'Web thk.', 'Flange width', and 'Flange thk.' are listed. A diagram of a T-beam cross-section is shown with dimensions b1, h, y, z, t, and b. An 'Existing sections:' dropdown menu is also available.

At the bottom right, a 3D perspective view of a girder is shown with labels for 'PLATE', 'STIFFENER', and 'GIRDER'. The 'Girder length (Lg)' is set to 10. A 'Save and return structure' button is located at the bottom right.

Define tanks

Tanks are searched for when clicking “Find compartments”. Non watertight structure are ignored. For information on structure types click “Show structure types”.

After tanks are found content and overpressure must be defined as seen next.

Find compartments

External pressures

Display current compartments

Comp. no.: **2**

2
3
4
5

Tank content : crude_oil

Tank density : 1025.0 [kg/m³]

Overpressure : 25000.0 [Pa]

Max elevation : 30.9

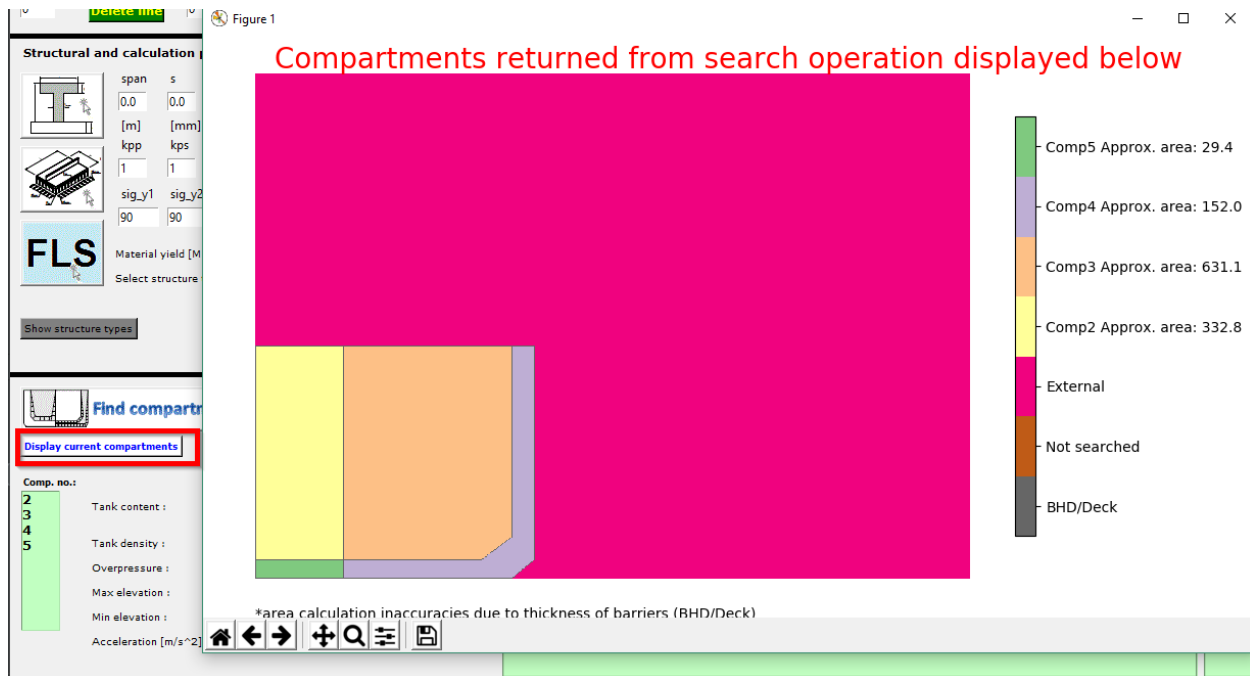
Min elevation : 2.5

Set compartment properties.

Delete all tanks

Accelerations [m/s²]:
static: 9.81 , dynamic loaded: 3.0 , dynamic ballast: 3.0

If you press “Display current compartments” after doing a compartment search, the result of the search is illustrated as seen next. Approximate area of the respective compartments is also shown.



Define external pressures

Click “External pressures” to define pressures acting on the structures.

NOTE:

FOR DYNAMIC EQUATION THE FOLLOWING APPLIES

X (horizontal) used for BOTTOM, BBT, HOPPER, MD

Y (vertical) used for BBS, SIDE_SHELL, SSS

After new window is opened:

1. Make dynamic loads

- Dynamic loads are made by defining up to 3rd degree equations. X or Y direction depends on the defined structure type.
- Note that you can define a constant dynamic load by using Constant (Constant (C)) only.

2. Static loads are calculated according to depth.

3. To apply a defined load to a line or multiple lines:
 - a. a. Select load by clicking the created load
4. Click the lines that shall have the load. Click the button “Press to add selected lines to selected load”
5. When finished press the button in the upper right corner.

Load properties

1. Dynamic loads
Define dynamic loads as an polynomial curve.
Can be third degree, second degree, linear or constant

Input load name:

Third degree poly [x^3]:

Second degree poly [x^2]:

First degree poly [x]:

Constant [C]:

Load condition:

Limit state: **Create dynamic load**

2. Static loads
Hydrostatic loads defined by draft.

Define name of static load: **Create static load**

Define static draft from sea:

Select load condition:

3. Slamming pressure

Load name:

Pressure [Pa]: **Create slamming load**

Press to add selected lines to selected load

Select a load in "1." to and then choose lines to apply to load
(select by clicking lines). Alternatively define manually ----->

ballast_side

House left click: select lines to loads
House right click: clear all selection
Right key click: add selected line
Control key press: remove selected line

**4. Created loads are seen below
(double click to select):**

Delete selected load

Select to see associated lines:

ballast_side	line50
ballast_bottom	line51
loaded_static	line52
ballast_static	line53
slamming	line54
loaded_bottom	line55

Properties selected load is:

Name of load: ballast_side
Polynomial (x^3): 0.0
Polynomial (x^2): 303.0
Polynomial (x): -3750.0
Constant (C): 153000.0
Load condition: ballast
Limit state: ULS
Is external?: True
Static draft: None

Load combinations

Load combinations are created automatically after external pressures are defined.
Some comments on the loads.

1. According to DNVGL-OS-C101
2. Highest pressure are chosen w.r.t. tank filling.

3. You can deselect a load by manually inputting load factor to 0 or deselect include.

Optimization

Optimization iteration by predefined stiffeners

From 0.5 you can iterate by a defined set of stiffeners. Press the button marked below. Open a csv (or json) file. Then start your iterations. The only other input is the stiffener spacing and plate thickness.

To see how the input format is click the “open predefined stiffeners example” button. See illustrations next.

Open predefined stiffeners example

RUN OPTIMIZATION!

anysmart

algorithm information

Iterate predefined stiffeners

Note that the weight of your initial structure is ignored even though it is calculated. If the initial structure is in your predefined set it will be included in the evaluations.

Press the button indicated below to activate. A open file window will open when running the optimization.

-- Structural optimizer --

Return and replace initial structure with optimized

Iterate predefined stiffeners

	Spacing [mm]	Plate thk. [mm]	Web height [mm]	Web thk. [mm]	Flange width [mm]	Flange thk. [mm]
Upper bounds [mm]	850.0	25.0	600.0	35.0	300.0	40.0
Iteration delta [mm]	50.0	2.0	50.0	2.0	50.0	2.0
Lower bounds [mm]	650.0	10.0	400.0	15.0	100.0	20.0

Estimated running time for algorithm: 7 seconds

RUN OPTIMIZATION!

Single optimization

Single optimization is done by clicking a line and clicking the “OPTIMIZE” button.

1. Set the upper and lower bounds of the optimization.
2. Set the delta to be used for the searched. This is the step size of the optimization when using brute force method (for example anysmart).
3. Run the optimization.
4. If you are happy, return the properties by clicking the top button

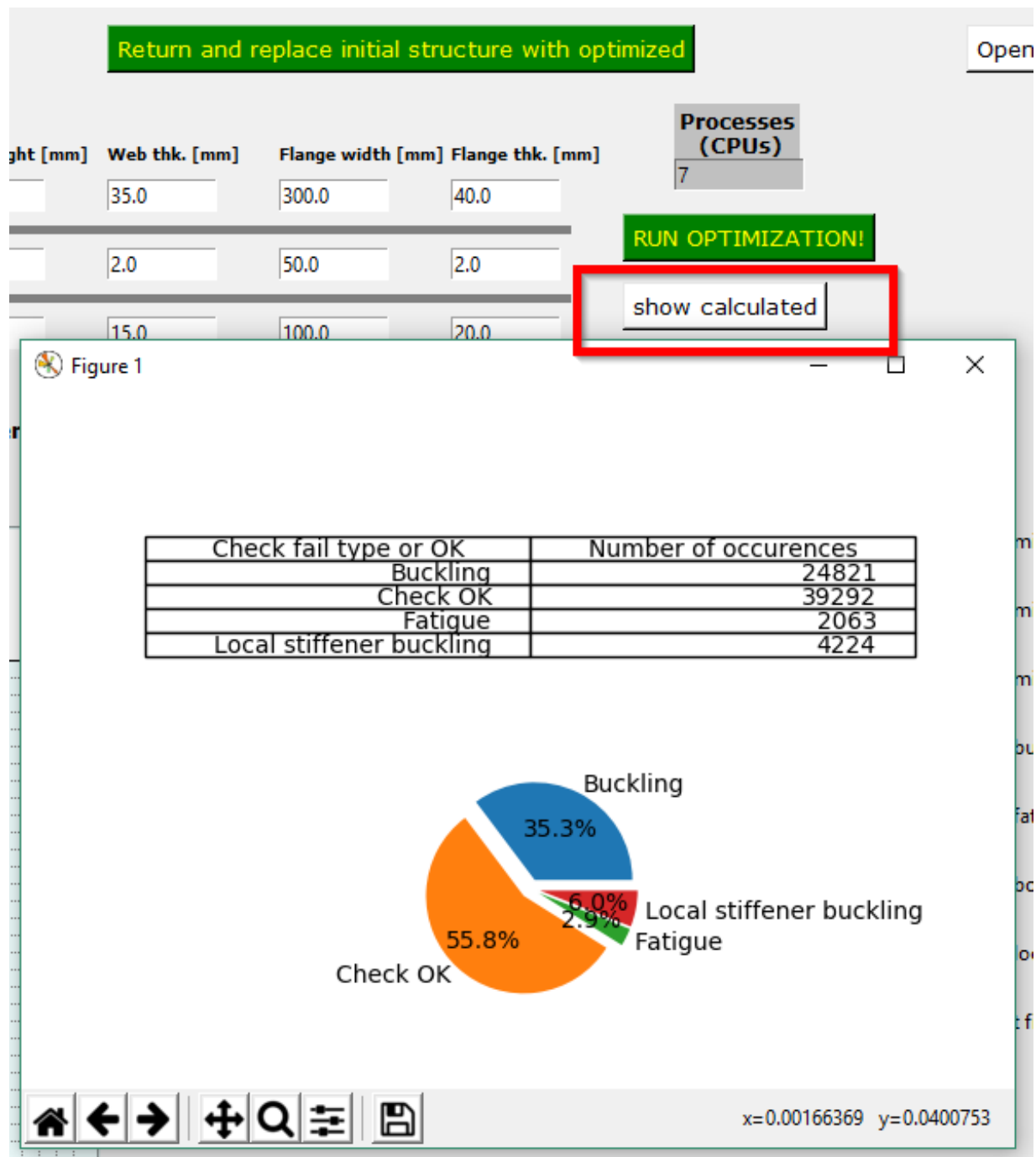
Various checks in the optimization module:

You can select the checks to be performed.

The weight filter ensures that only sections with a lower weight than the current minimum weight. This significantly speed up the calculations, but if you want to see the full distribution of the various checks this must be unchecked.

Check for minimum section modulus	<input checked="" type="checkbox"/>
Check for minimum plate thk.	<input checked="" type="checkbox"/>
Check for minimum shear area	<input checked="" type="checkbox"/>
Check for buckling (RP-C201)	<input checked="" type="checkbox"/>
Check for fatigue (RP-C203)	<input checked="" type="checkbox"/>
Check for bow slamming	<input type="checkbox"/>
Check for local stf. buckling	<input checked="" type="checkbox"/>
Use weight filter (for speed)	<input checked="" type="checkbox"/>

If you press the “show calculated” button, you will get an overview of how many is ok and how many failed (and what criteria first failed). One “occurence” is a one checked plate/stiffener combination.



You will also be asked to save to a csv file. If you do not cancel, a csv file will ALL results will pre saved to your chosen location. If you open the file in excel you should see something like show next

Multiple optimization

Multiple optimization is done by clicking the “MultiOpt” button.

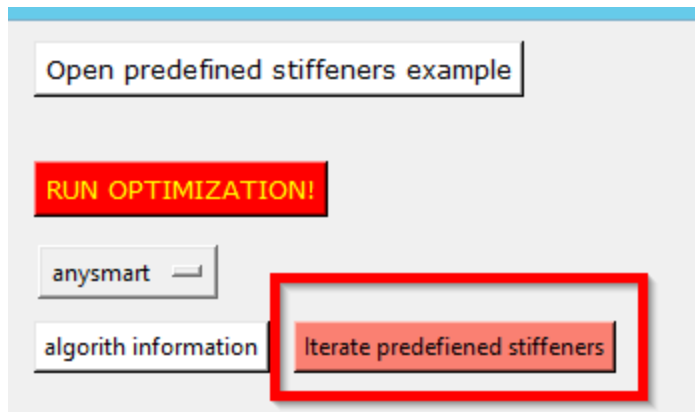
1. Same input on upper bounds, lower bounds and delta.
2. Click all the lines you want to include in the optimization.
3. Run the optimization.
4. Check the properties by **middle clicking** the line you ran.
5. If you are happy return the properties by clicking the top button

Other options that can be set is explained in the single optimization chapter.

When showing calculated you must have selected a line (middle click).

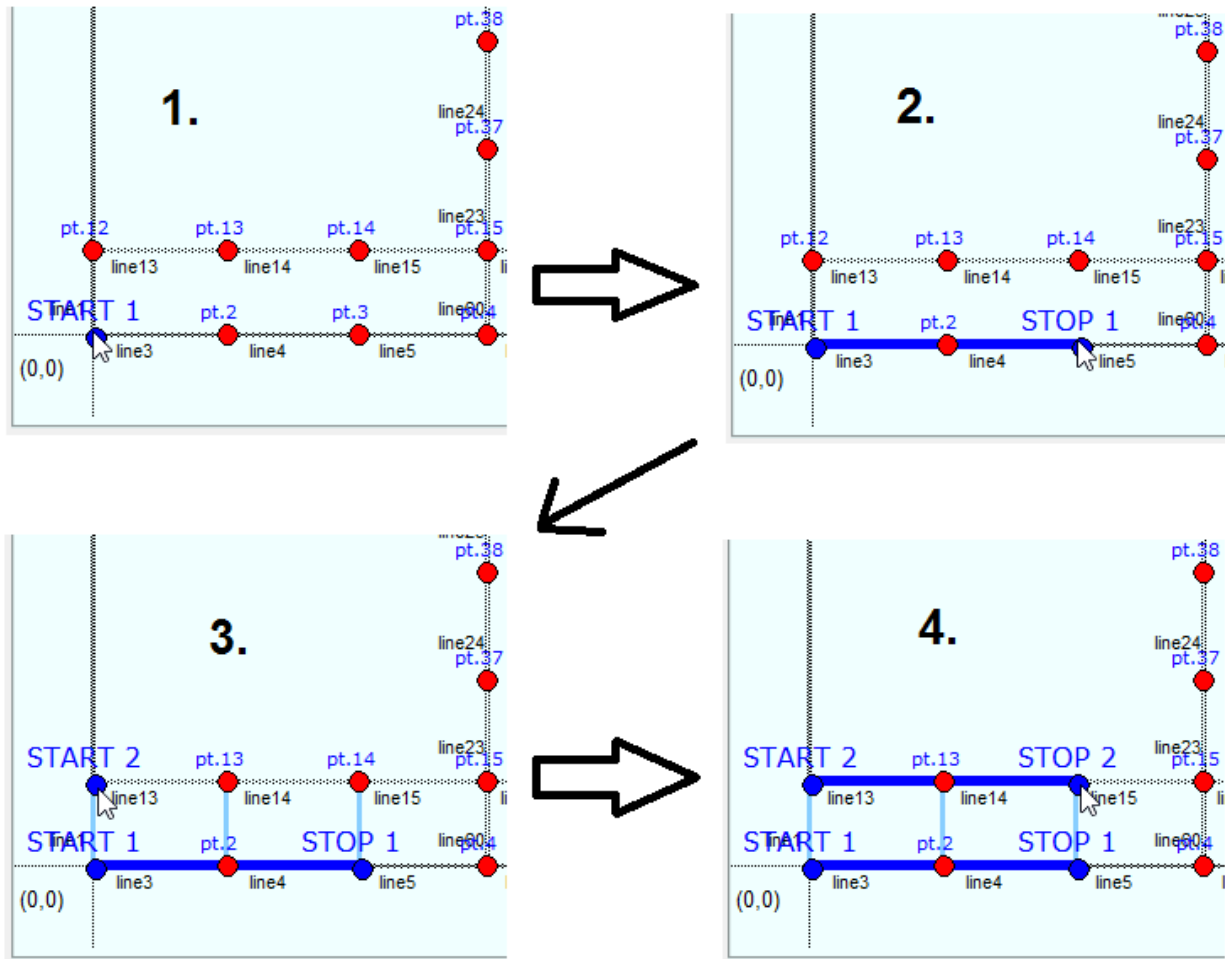
Span optimization

NOTE: The span optimization is computationally heavy. It is recommended to use a set of predefined stiffeners.



The optimization is started as follows.

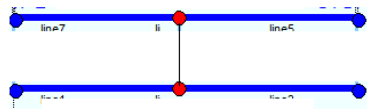
1. Start by clicking as illustrated next:



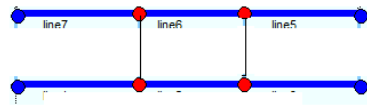
2. Then run optimization.

The program will calculate variations of even spans in your structure as illustrated next. This is an example and number of plate fields may vary.

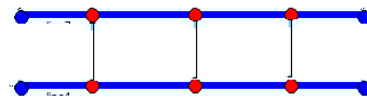
4 plate fields



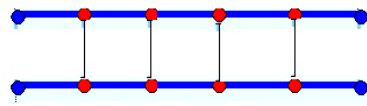
6 plate fields



8 plate fields



10 plate fields



You can, similar to single optimization, select the checks that shall be runned. Also you can set the girder (frame) properties. This is used for calculating the weights.

With reference to the example above, max span mult is the multiplier for the 4 plate fields set up and min span mult is the weight multiplication for the 10 plate field set up. This is adopted because one can assume the required dimensions for the girder will reduce when more girders are added.

Minimum span and maximum span is the minimum and maximum span of the plate fields in meters.

Check for minimum section modulus	<input checked="" type="checkbox"/>	Frame (girder data) for weight calculation:	
Check for minimum plate thk.	<input checked="" type="checkbox"/>	Girder thickness	<input type="text" value="0.018"/>
Check for minimum shear area	<input checked="" type="checkbox"/>	Stiffener height	<input type="text" value="0.25"/>
Check for buckling (RP-C201)	<input checked="" type="checkbox"/>	Stiffener thickness	<input type="text" value="0.015"/>
Check for fatigue (RP-C203)	<input checked="" type="checkbox"/>	Stf. flange width	<input type="text" value="0"/>
Check for bow slamming	<input checked="" type="checkbox"/>	Stf. flange thickenss	<input type="text" value="0"/>
Check for local stf. buckling	<input checked="" type="checkbox"/>	For weight calculation of girder: Max span mult / Min span mult	
		<input type="text" value="1.2"/>	<input type="text" value="0.8"/>
		Maximum span / Minimum span ->	<input type="text" value="6"/> <input type="text" value="2"/>

Results are presented as seen next.

RUN OPTIMIZATION!

anysmart

algorithm information

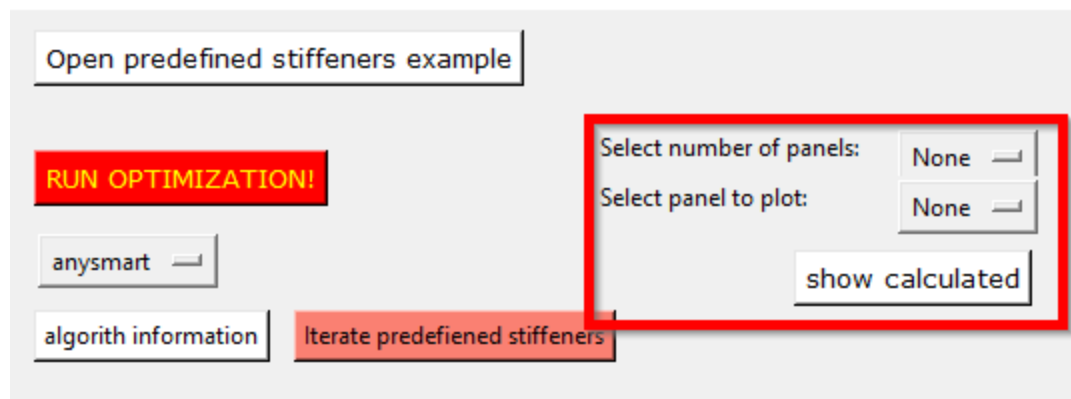
Results seen next. Weight index is tot_weight / max_weight
max_weight is the highest total weight of the checked variations.
Weight index of 1 is the heaviest calculated variation.

Plate fields	Fields length	Weight index	All OK?

4	6.0	1.0	True
6	4.0	0.768	True
8	3.0	0.765	True
10	2.4	0.825	True

In this case 8 plate fields with length of 3 meter will give the lowest weight. 6 plate fields is almost equal.

When the analysis has been runned you should save your results. Just specify a file name in the save file dialog. You can also get detailed individual results for a specified panel. Select number of plate fields in the iteration you want to look at, then choose which panel to get data from. Order of the panels is the same as printed in the left result canvas.



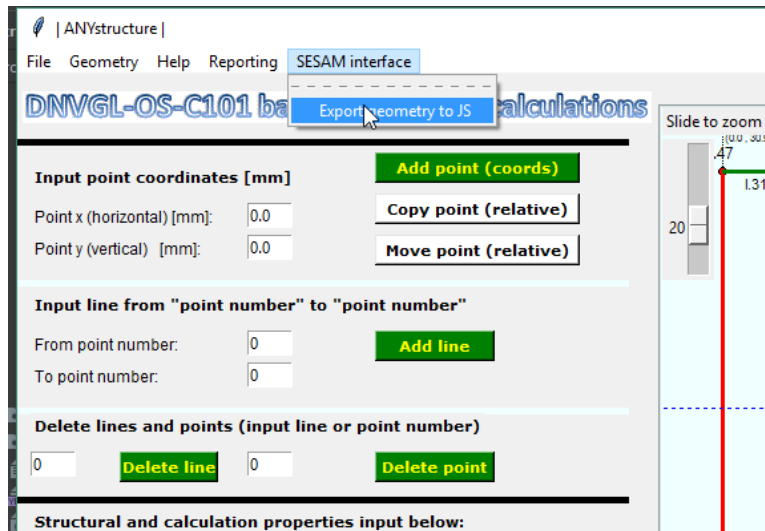
Now close the window. Results are not currently returned to main window.

Detailed results, printed after running, looks like this :

[illegible]

Export to JS

ANYstructure can export points, lines and section properties to SESAM GeniE. A dialog will request a location to save the JS file. After that you can read the js file into GeniE.



The result is illustrated below:

