

ANYstructure documentation

Input point coordinates [mm]

Point x (horizontal) [mm]: 0.0
Point y (vertical) [mm]: 0.0

Input line from "point number" to "point number"

From point number: 0
To point number: 0

Delete lines and points (input line or point number)

9
Delete line
Delete point

Structural and calculation properties input below:

span s pt_thk web_h web_thk fl_w fl_thk
3.0 700.0 18.0 400.0 12.0 200.0 18.0
[m] [mm] [mm] [mm] [mm] [mm] [mm]

kpp kps km1 km2 k3
1.0 1.0 12.0 24.0 12.0

sig_y1 sig_y2 sig_x tau_y1 off type pressure side
100.0 100.0 100.7 5.0 IT p

Material yield [MPa]: 355.0
Select structure type: BOTTOM
Internal pressure from comp.

Find compartments **External pressures**

Display current compartments:

Comp. no.: 2, 3, 4, 5

Tank contents: []
Tank density: 1025 [kg/m³]
Overpressure: 23000 [Pa]
Max elevation: 0.0
Min elevation: 0.0
Acceleration [m/s²]:

Static and dynamic accelerations line9

Static acceleration (m/s²): 0.81
Dyn. acc. loaded (m/s²): 3.0
Dyn. acc. hullast (m/s²): 3.0

Optimize selected line/structure (right click line):
OPTIMIZE MultiOpt SPAN

Combination for line (select line). Change with slider:
OS-C101 Table 1: DNV a) 2: DNV b) 3: TankTest

Section moduli: Wey1: 4.7000E+06 [mm³], Wey2: 1.9600E+06 [mm³]
Minimum section modulus: 3.9421E+06 [mm³]
Shear area: 5.2320E+03 [mm²]
Minimum shear area: 3.7877E+03 [mm²]
Plate thickness: 18.0 [mm]
Minimum plate thickness: 14.7 [mm]

Buckling results DNV RP C201 (z* optimized):
[eq 2.19]: 0.83 [eq 2.50]: 0.92 [eq 7.53]: -0.13 [7.52]: 0.62 [eq 7.53]: 0.92 [z*]: 0.14

Fatigue results (DNVGL RP C203):
Total damage (DFF not included): 0.03 | With DFF = 2.0 -> Damage: 0.06

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 shows a software interface for defining structure properties. The main window is titled "Define structure properties" and contains several sections:

- Input line from "point number" to "point number"**: Fields for "From point number" (0) and "To point number" (4), with "Add line" and "Delete line" buttons.
- Structural and calculation properties input below:** A table for inputting properties for a beam section. The table has columns for span, plate thickness, web height, web thickness, flange width, and flange thickness, with units in meters and millimeters. It also includes fields for material yield strength, buckling properties, and fatigue properties.
- Define plate and stiffener properties:** A section with input fields for Plate thk., Web height, Web thk., Flange width, and Flange thk., along with a "type" dropdown menu.
- Existing sections:** A diagram showing various beam section profiles with dimensions labeled (b1, h, y, z, etc.). A red box highlights this section with the text "Existing sections".
- Define buckling calculation properties:** A section with a "Material yield [MPa]" field (355.0) and a "Select structure type" dropdown menu (BOTTOM).
- Define fatigue properties:** A section with "Tank content" (2, 3, 4, 5), "Tank density" (1025), and "Overpressure" (25000) fields.
- Summary:** A section displaying the defined properties: "Plate: 700.0x18.0", "Web: 400.0x12.0", and "Flange: 250.0x14.0".
- Diagram:** A 3D perspective view of a beam section with labels for "PLATE", "GIRDER", and "STIFFENER". The "Girder length (Lg)" is set to 10.
- Buttons:** "Add structure to line" (green), "Save and return structure" (green), and "Delete all tanks" (red).

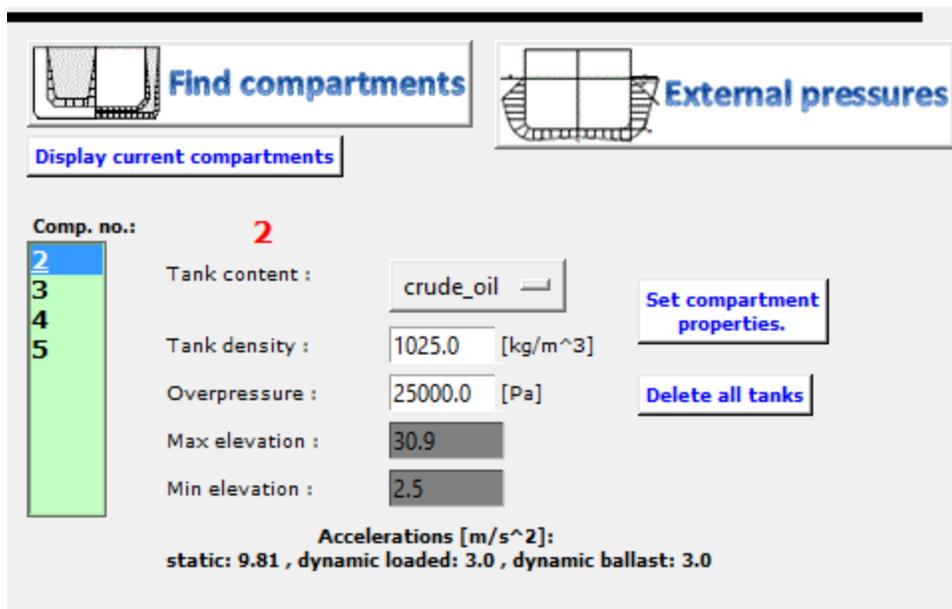
Red callout boxes with arrows point to the following elements:

- "Define plate and stiffener properties." (points to the input fields for plate and web properties)
- "Existing sections" (points to the diagram of various beam profiles)
- "Define buckling calculation properties." (points to the material yield and structure type fields)
- "Define fatigue properties." (points to the tank content and density fields)

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.

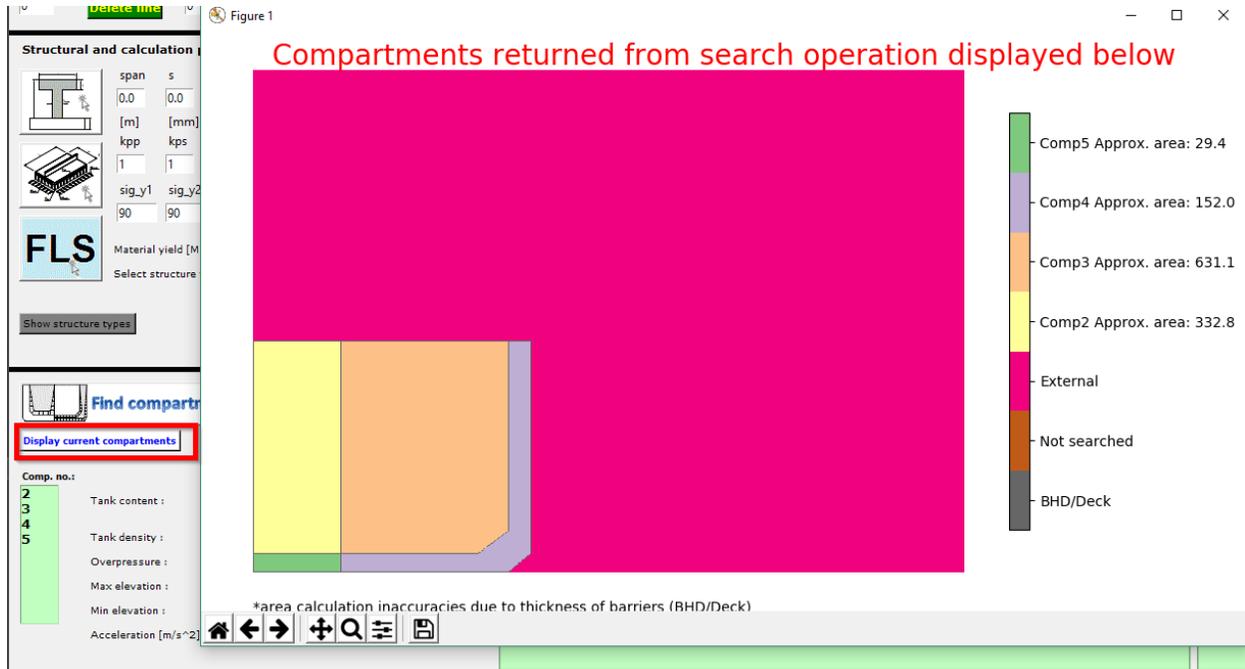


The screenshot shows a software interface with two main sections: "Find compartments" and "External pressures". Below these is a button "Display current compartments". A list of compartments is shown on the left, with "Comp. no.:" and a red "2" above it. The list contains numbers 2, 3, 4, and 5, with 2 highlighted in blue and 3, 4, and 5 in green. To the right of the list are input fields for "Tank content" (set to "crude_oil"), "Tank density" (1025.0 [kg/m^3]), "Overpressure" (25000.0 [Pa]), "Max elevation" (30.9), and "Min elevation" (2.5). There are buttons for "Set compartment properties." and "Delete all tanks.". At the bottom, it shows "Accelerations [m/s^2]: static: 9.81 , dynamic loaded: 3.0 , dynamic ballast: 3.0".

Comp. no.:	Tank content :	Tank density :	Overpressure :	Max elevation :	Min elevation :
2	crude_oil	1025.0 [kg/m ³]	25000.0 [Pa]	30.9	2.5

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:

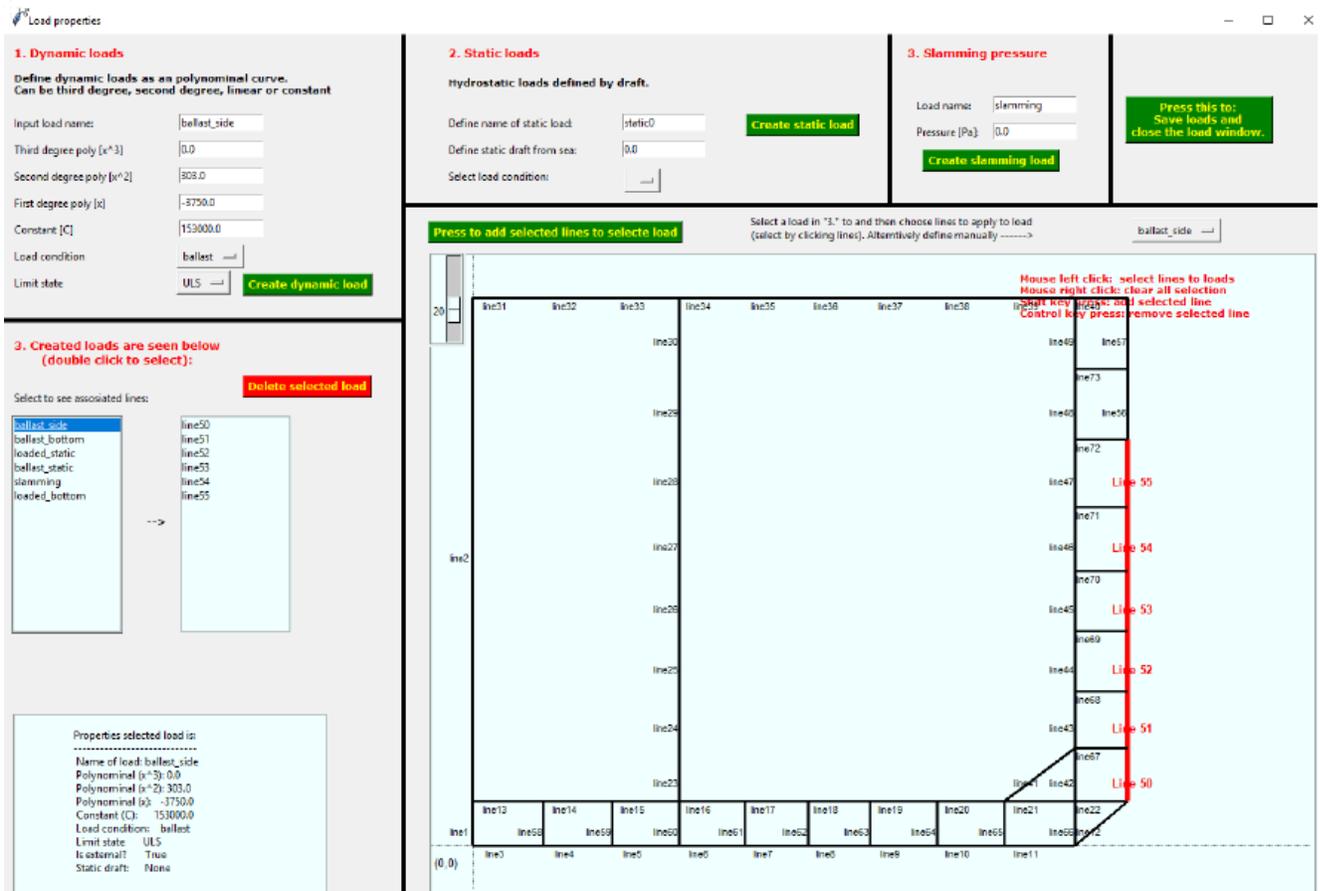
1. Make dynamic loads

a. Dynamic loads are made by defining up to 3rd degree equations. X or Y direction depends on the defined structure type.

b. 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 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.

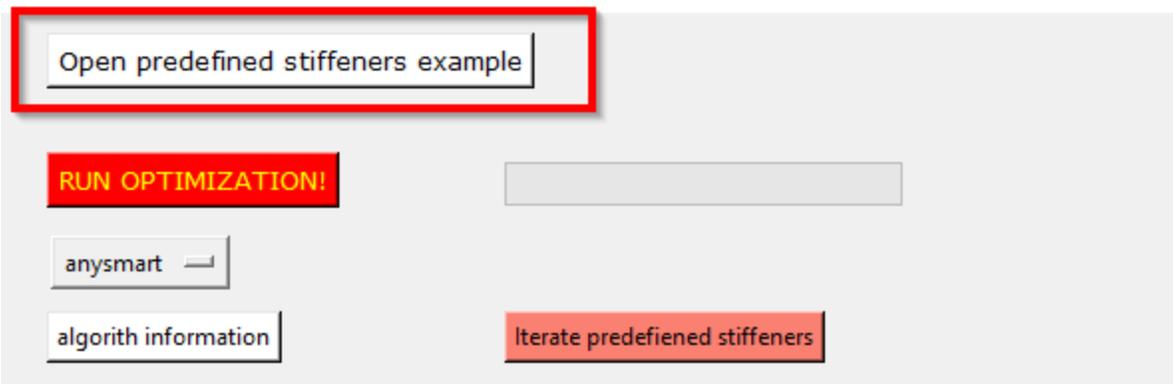
3. You can deselect a load by manually inputting load factor to 0 or deselect include.

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.



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

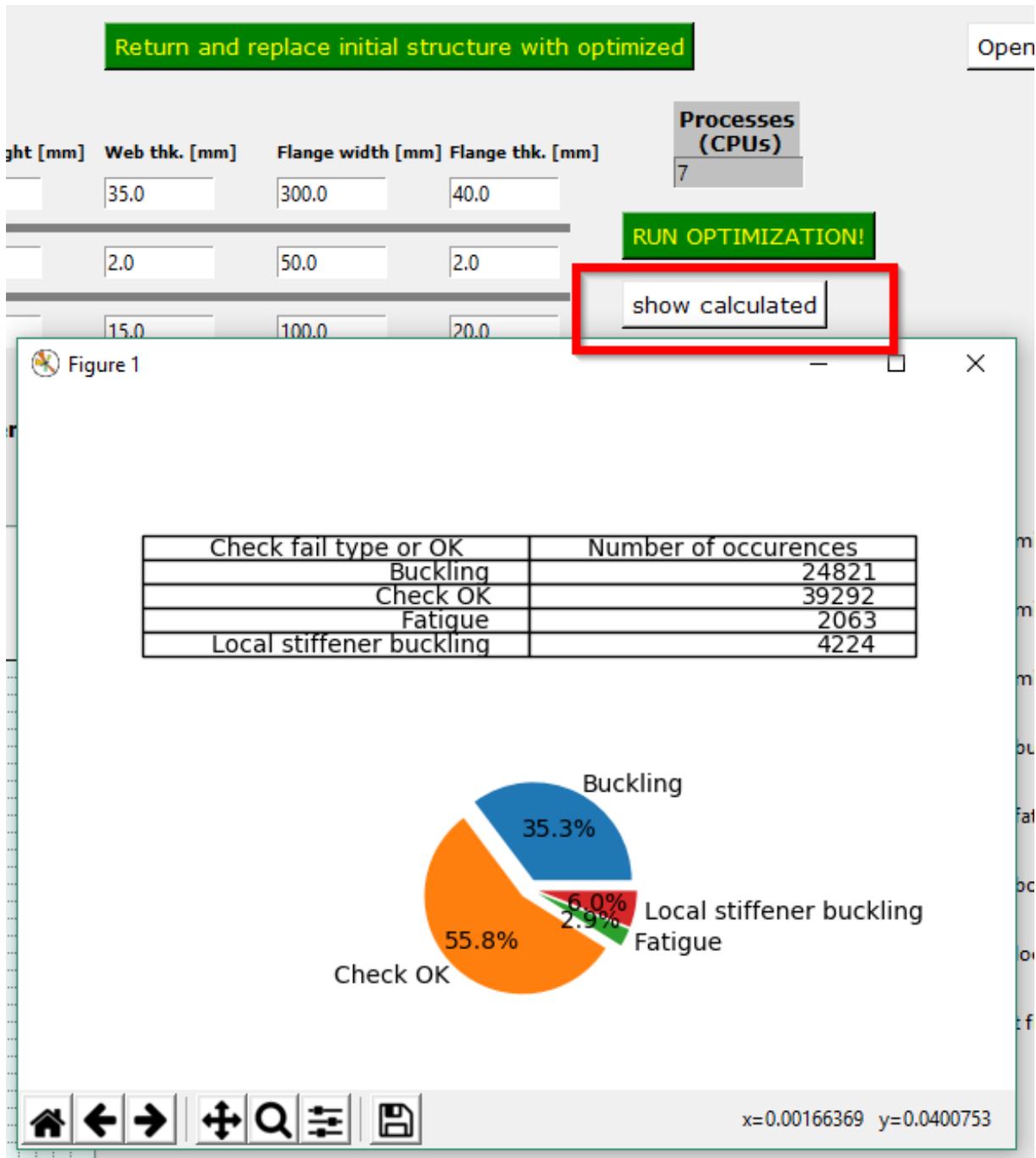
Various checks in the optimization module:

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 “occurrence” 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.

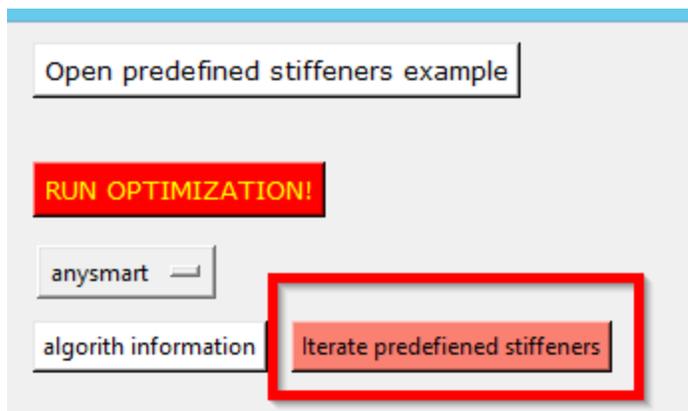
1. Same input on upper bounds, lower bounds and delta.
2. Click all the lines you want to include in the optimization.
3. Run the optimization.
4. Check the properties by **middle clicking** the line you ran.
5. If you are happy return the properties by clicking the top button

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

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

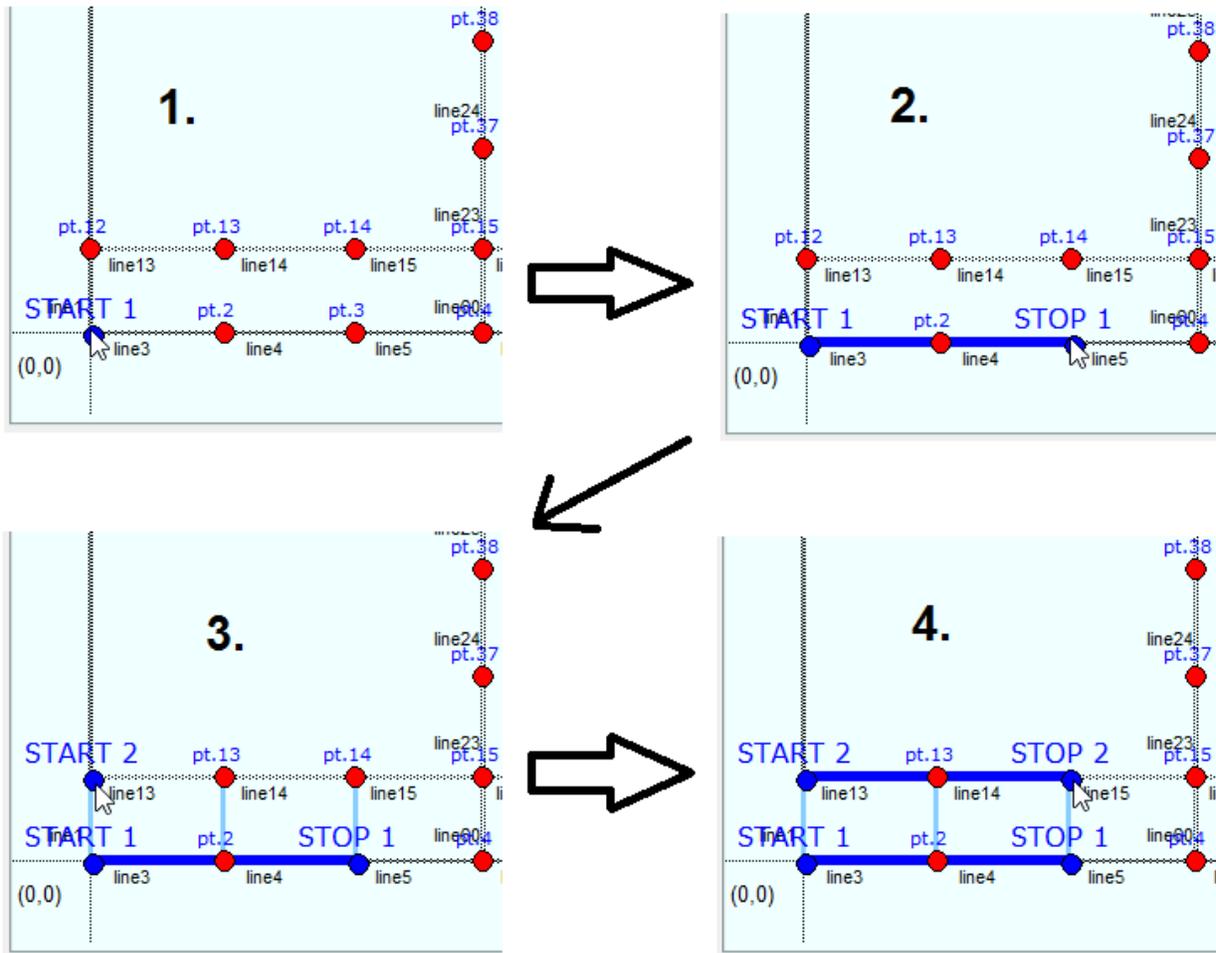
Span optimization

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



The optimization is started as follows.

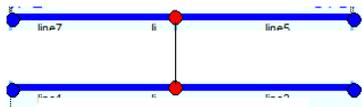
1. Start by clicking as illustrated next:



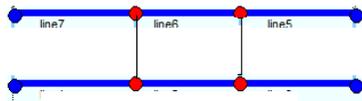
2. Then run optimization.

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.

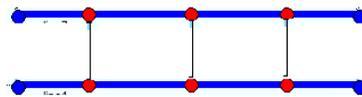
4 plate fields



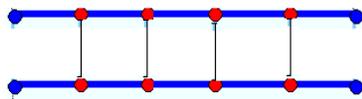
6 plate fields



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 thickness	<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!

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.

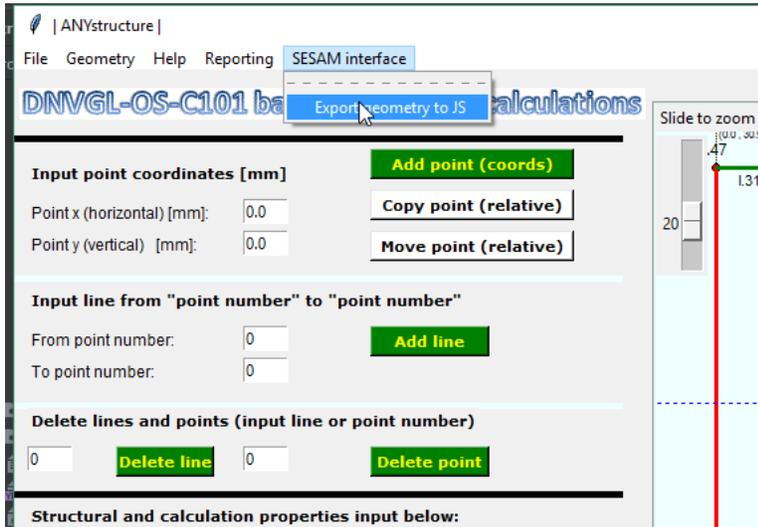
Plate fields	Fields length	Weight index	All OK?

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.

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:

