

# ANYstructure documentation

The screenshot displays the ANYstructure software interface, which is used for structural analysis. The interface is divided into several functional areas:

- Input point coordinates [mm]:** Includes fields for Point x (horizontal) [mm] (0.0), Point y (vertical) [mm] (0.0), and buttons for "Add point (coords)", "Copy point (relative)", and "Move point (relative)".
- Input line from "point number" to "point number":** Includes fields for "From point number:" (0) and "To point number:" (0), with an "Add line" button.
- Delete lines and points (or left/right click and use "Delete key"):** Includes fields for "Line number (left click):" (43) and "Point number (right click):" (0), with "Delete line" and "Delete point" buttons.
- Structural and calculation properties input below:** A table for defining structural properties:

span	s	pl_thk	web_h	web_thk	fl_w	fl_thk
0.0	0.0	0.0	0.0	0.0	0.0	0.0
[m]	[mm]	[mm]	[mm]	[mm]	[mm]	[mm]

Additional properties include kpp, kps, km1, km2, k3, sig\_y1, sig\_y2, sig\_x, tau\_y1, and stf type. Material yield [MPa] is set to 355, and Pressure side (p-plate, s-stf) is set to p.
- Select structure type:** Includes a dropdown menu set to "BOTTOM" and a "Show structure types" button.
- Find compartments** and **External pressures** sections are also visible.
- Static and dynamic accelerations:** Includes input fields for Static acceleration [m/s<sup>2</sup>] (9.81), Dyn. acc. loaded [m/s<sup>2</sup>] (3.0), and Dyn. acc. ballast [m/s<sup>2</sup>] (3.0), with a "Set accelerations" button.
- Optimize selected line/structure (right click line):** Includes buttons for "OPTIMIZE", "MultiOpt", and "SPAN".
- Combination for line [select line]:** Includes a table for defining combinations:

OS-C101 Table 1	1: DNV a)	2: DNV b)	3: TankTest
1			

The central visualization area shows a structural model of a ship hull with various points (pt.1 to pt.20) and lines (l.1 to l.19) defined. The model is color-coded (red, green, blue) and includes labels for "loaded\_static [m]" and "ballast\_static [m]".

January 2020

Version 1.0

# Table of contents

<b>Table of contents</b>	<b>2</b>
<b>Theory</b>	<b>3</b>
<b>Modelling</b>	<b>4</b>
<b>Assigning properties</b>	<b>5</b>
Display properties	6
<b>Define tanks</b>	<b>6</b>
<b>Setting accelerations</b>	<b>8</b>
<b>Define external pressures</b>	<b>8</b>
<b>Load combinations</b>	<b>9</b>
<b>Results</b>	<b>10</b>
<b>Optimization</b>	<b>11</b>
Optimization iteration by predefined stiffeners	11
Single optimization	12
Multiple optimization	14
Span optimization	14
<b>Reporting</b>	<b>19</b>
<b>Export to JS</b>	<b>20</b>

# Theory

All calculations are according to the following DNVGL standards and recommended practices:

- DNVGL-OS-C101 Design of offshore steel structures, general - LRFD method
  - <http://rules.dnvgl.com/docs/pdf/DNVGL/OS/2018-07/DNVGL-OS-C101.pdf>
- DNV-RP-C203 Fatigue design of offshore steel structures
- DNV-RP-C201 BUCKLING STRENGTH OF PLATED STRUCTURES
  - <https://rules.dnvgl.com/docs/pdf/DNV/codes/docs/2010-10/RP-C201.pdf>



DNV-GL

# 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)

<b>Input point coordinates [mm]</b>	<input type="text"/>	<b>Add point (coords)</b>
Point x (horizontal) [mm]:	<input type="text" value="0.0"/>	<b>Copy point (relative)</b>
Point y (vertical) [mm]:	<input type="text" value="0.0"/>	<b>Move point (relative)</b>
<b>Input line from "point number" to "point number"</b>		
From point number:	<input type="text" value="0"/>	<b>Add line</b>
To point number:	<input type="text" value="0"/>	
<b>Delete lines and points (or left/right click and use "Delete key")</b>		
Line number (left click):	<input type="text" value="43"/>	<b>Delete line</b>
Point number (right click):	<input type="text" value="0"/>	<b>Delete point</b>

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
- DELETE** Delete selected line/point

# 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. If you have added a property to a line and want to use the same for the next line, just press “Add structure to line” on the new line.

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 a "Define structure properties here --" section. This section includes a "type:" dropdown menu set to "T", a table of dimensions (Plate thk., Web height, Web thk., Flange width, Flange thk.), and a diagram of a T-beam section with dimensions labeled (b1, h, tw, tf, b, z, y, e, z, Fb hxt). A red box labeled "Existing sections" points to a dropdown menu. Below this, a table lists the defined properties: Plate: 700.0x18.0, Web: 400.0x12.0, Flange: 250.0x14.0. To the right, a 3D perspective view of a girder with a plate and stiffener is shown, with labels for "PLATE", "STIFFENER", and "GIRDER", and a dimension for "Girder length (Lg)" set to 10. A green button at the bottom right says "Save and return structure".

On the left side of the interface, there are several panels:

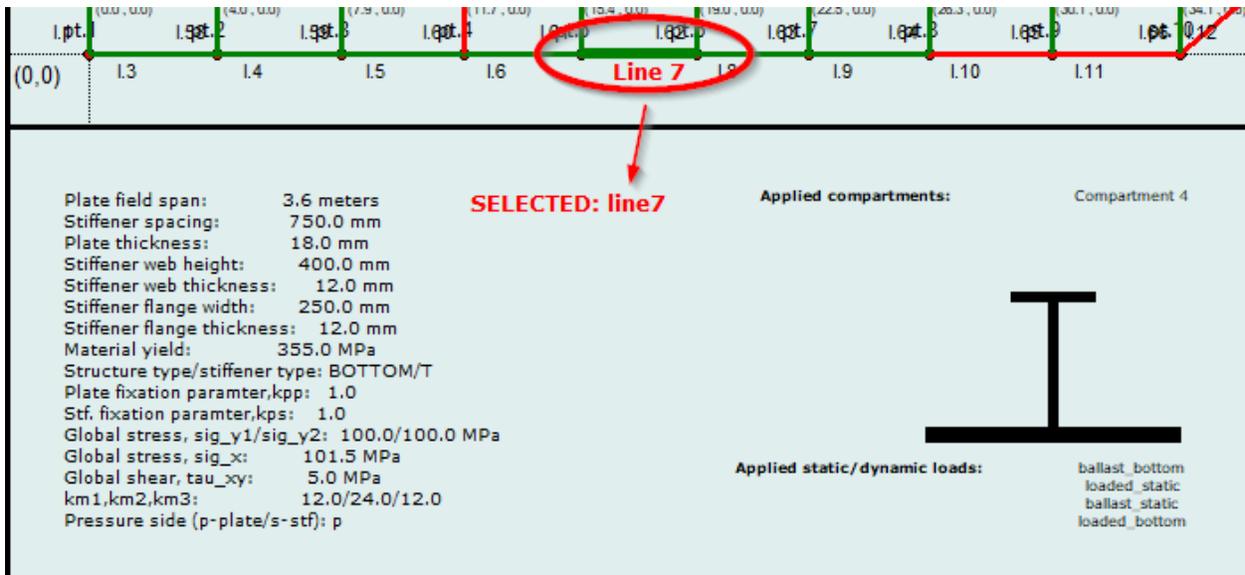
- Input line from "point number" to "point number"**: Includes fields for "From point number:" (0) and "To point number:" with an "Add line" button.
- Delete lines and points**: Includes a "Delete line" button.
- Structural and calculation properties input below:** A table with columns for span, plate thickness, web height, web thickness, flange width, and flange thickness, with units in meters and millimeters. It also includes fields for material yield strength (355.0 MPa) and structure type (BOTTOM).
- Find compartments**: A section for defining compartments, including "Comp. no.", "Tank content", "Tank density", and "Overpressure".

Red callout boxes highlight specific features:

- "Define plate and stiffener properties." points to the structural input table.
- "Define buckling calculation properties." points to the "Find compartments" section.
- "Define fatigue properties." points to the "Add structure to line" button.

## Display properties

If you click a line properties is displayed in the window below as seen next.



The screenshot shows a coordinate system at the top with points labeled 1.pt. through 12.pt. along a horizontal axis. A red circle highlights 'Line 7' between points 1.9 and 1.8. Below this, a properties window is displayed with the following information:

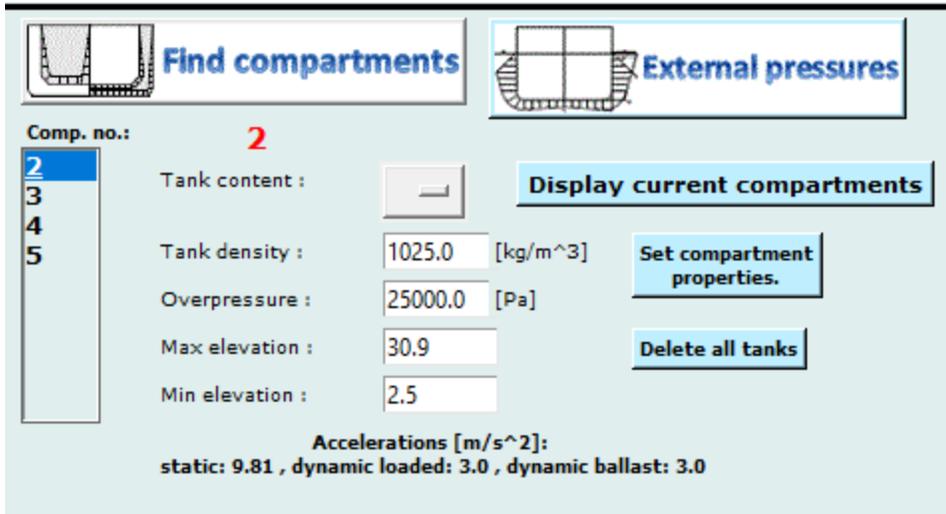
Plate field span:	3.6 meters	<b>SELECTED: line7</b>	Applied compartments:	Compartment 4
Stiffener spacing:	750.0 mm			
Plate thickness:	18.0 mm			
Stiffener web height:	400.0 mm			
Stiffener web thickness:	12.0 mm			
Stiffener flange width:	250.0 mm			
Stiffener flange thickness:	12.0 mm			
Material yield:	355.0 MPa			
Structure type/stiffener type:	BOTTOM/T			
Plate fixation paramter,kpp:	1.0			
Stf. fixation paramter,kps:	1.0			
Global stress, sig_y1/sig_y2:	100.0/100.0 MPa			
Global stress, sig_x:	101.5 MPa			
Global shear, tau_xy:	5.0 MPa		Applied static/dynamic loads:	ballast_bottom
km1,km2,km3:	12.0/24.0/12.0			loaded_static
Pressure side (p-plate/s-stf):	p			ballast_static
				loaded_bottom

A diagram of a T-shaped stiffener is shown on the right side of the properties window.

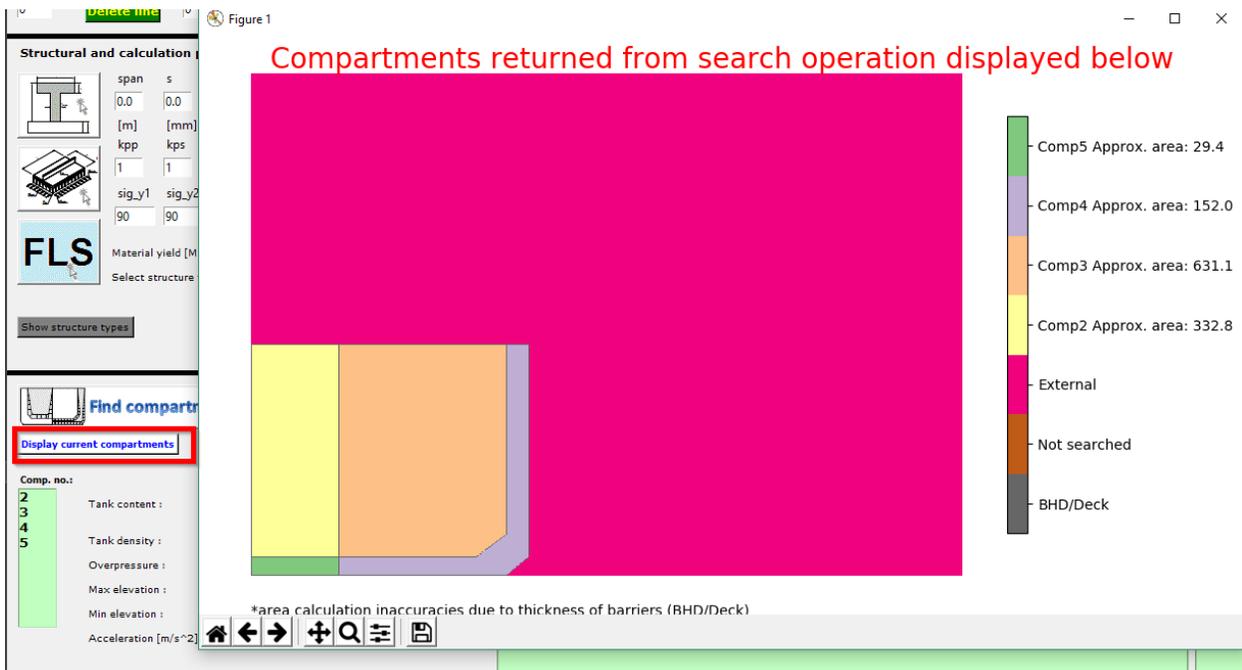
## Define tanks

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

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

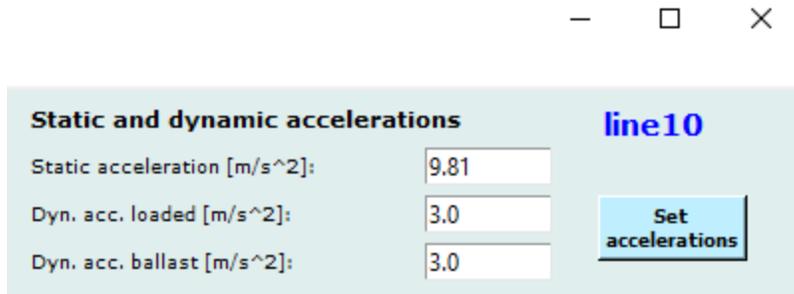


If you press “Display current compartments” after doing a compartment search, the result of the search is illustrated as seen next. Approximate area of the respective compartments is also shown.



## Setting accelerations

Accelerations applies to tank content. It is set in the upper right corner as seen next.



Static and dynamic accelerations **line10**

Static acceleration [m/s<sup>2</sup>]:

Dyn. acc. loaded [m/s<sup>2</sup>]:

Dyn. acc. ballast [m/s<sup>2</sup>]:

**Set accelerations**

## 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**

- Click the lines that shall have the load. Click the button “Press to add selected lines to selected load”
- When finished press the button in the upper right corner.

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

Input load name: ballast\_side  
 Third degree poly [x^3]: 0.0  
 Second degree poly [x^2]: 303.0  
 First degree poly [x]: -3750.0  
 Constant [C]: 153000.0  
 Load condition: ballast  
 Limit state: ULS  
 Create dynamic load

**2. Static loads**  
 Hydrostatic loads defined by draft.

Define name of static load: static0  
 Create static load  
 Define static draft from sea: 0.0  
 Select load condition: [dropdown]  
 Create slamming load

**3. Slamming pressure**

Load name: slamming  
 Pressure [Pa]: 0.0  
 Create slamming load

Press this to: Save loads and close the load window.

Press to add selected lines to selected load  
 Select a load in "3." to and then choose lines to apply to load (select by clicking lines). Alternatively define manually -----> ballast\_side

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

**3. Created loads are seen below**  
 (scroll if not all is shown.)  
 DOUBLE CLICK load to see associated lines.  
 Delete selected load

Select to see associated lines:

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

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.

- According to DNVGL-OS-C101
- Highest pressure are chosen w.r.t. tank filling.
- You can deselect a load by manually inputting load factor to 0 or deselect include.

Combination for line (select line). Change with slider.:

OS-C101 Table 1 1: DNV a) 2: DNV b) 3: TankTest

1

Name:	Stat LF	Dyn LF	Include?
ballast_bottom	0.0	0.7	<input checked="" type="checkbox"/>
loaded_static	1.3	0.0	<input checked="" type="checkbox"/>
ballast_static	1.3	0.0	<input checked="" type="checkbox"/>
loaded_bottom	0.0	0.7	<input checked="" type="checkbox"/>
Compartment4	1.2	0.7	<input checked="" type="checkbox"/>
Manual (pressure/LF)	0.0	1.0	<input checked="" type="checkbox"/>

Pressures for this line:  
(DNV a/b [loaded/ballast], tank test, manual)  
Note that ch. 4.3.7 and 4.3.8 is accounted for.

DNV a [Pa]: [462698, 248632]      DNV b [Pa]: [546435, 248430]  
TT [Pa]: [335707]                      Manual [Pa]: [0.0]

## Results

When clicking a line, results as presented in the window below. If the result for the clicked line is OK, the color of the line and text is green. If the result is NOT OK, the color of the line and text is red. Two examples are seen next.

All results ok

Section modulus: Wey1: 4.8300E+06 [mm<sup>3</sup>], Wey2: 1.7500E+06 [mm<sup>3</sup>]  
Minimum section modulus: 1.7163E+06 [mm<sup>3</sup>]  
Shear area: 5.1600E+03 [mm<sup>2</sup>]  
Minimum shear area: 3.5296E+03 [mm<sup>2</sup>]  
Plate thickness: 18.0 [mm]  
Minimum plate thickness: 15.1 [mm]  
Buckling results DNV-RP-C201 (z\* optimized):  
|eq 7.19: 0.88 |eq 7.50: 0.92 |eq 7.51: -0.19 |7.52: 0.6|eq 7.53: 0.92 |z\*: 0.12  
Fatigue results (DNVGL-RP-C203):  
Total damage: NO RESULTS

Section modulus not ok  
Buckling not ok

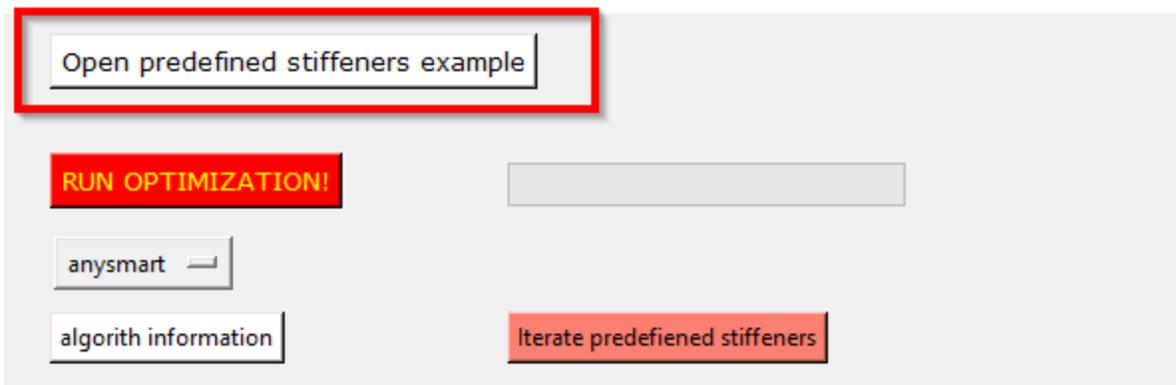
Section modulus: Wey1: 4.2400E+06 [mm<sup>3</sup>], Wey2: 1.4700E+06 [mm<sup>3</sup>]  
Minimum section modulus: 2.0739E+06 [mm<sup>3</sup>]  
Shear area: 4.6560E+03 [mm<sup>2</sup>]  
Minimum shear area: 4.1297E+03 [mm<sup>2</sup>]  
Plate thickness: 18.0 [mm]  
Minimum plate thickness: 15.8 [mm]  
Buckling results DNV-RP-C201 (z\* optimized):  
|eq 7.19: 0.9 |eq 7.50: 1.39 |eq 7.51: 0.35 |7.52: 0.81|eq 7.53: 0.73 |z\*: 0.13  
Fatigue results (DNVGL-RP-C203):  
Total damage (DFF not included): 0.058 | With DFF = 2.0 --> Damage: 0.117

# Optimization

## Optimization iteration by predefined stiffeners

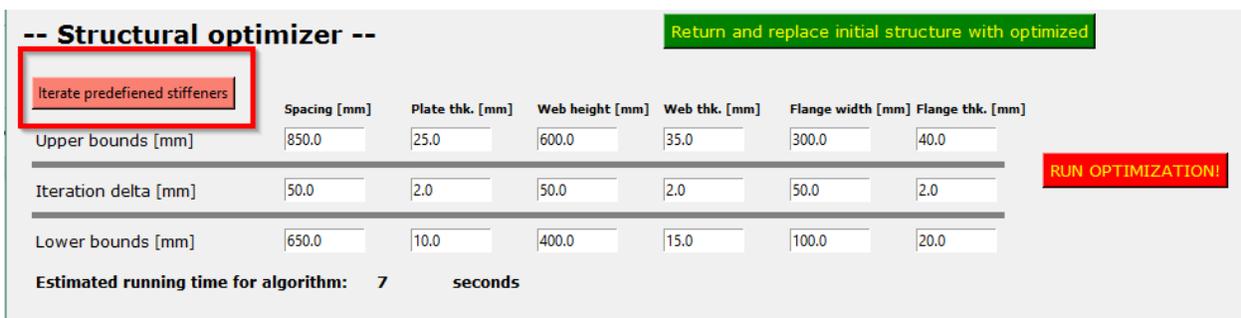
From 0.5 you can iterate by a defined set of stiffeners. Press the button marked below. Open a csv (or json) file. Then start your iterations. The only other input is the stiffener spacing and plate thickness.

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.



## 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

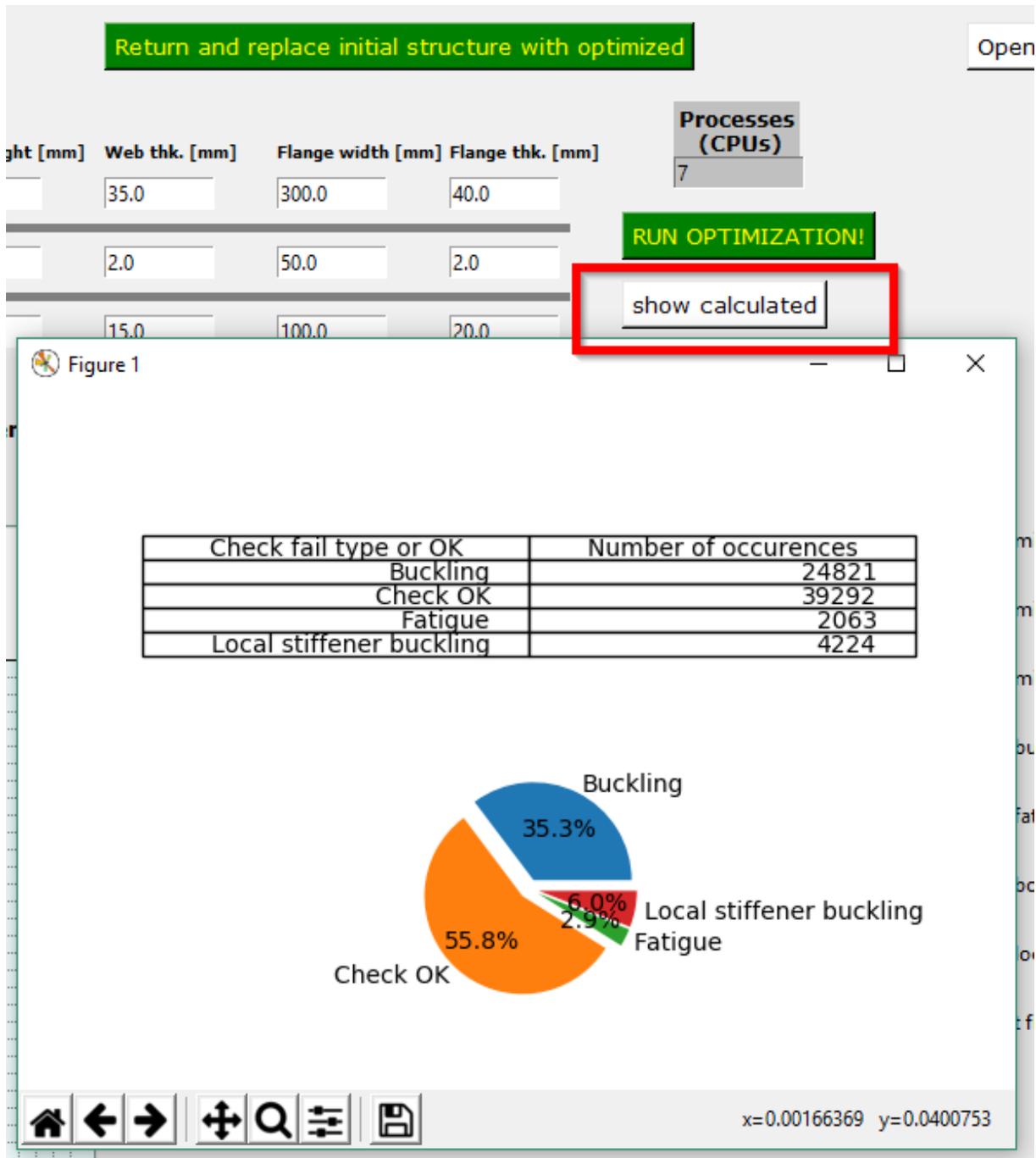
Various checks in the optimization module:

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.

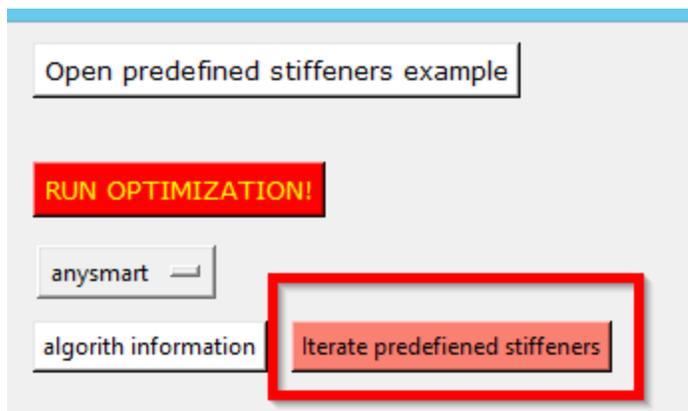
1. Same input on upper bounds, lower bounds and delta.
2. Click all the lines you want to include in the optimization.
3. Run the optimization.
4. Check the properties by **middle clicking** the line you ran.
5. If you are happy return the properties by clicking the top button

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

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

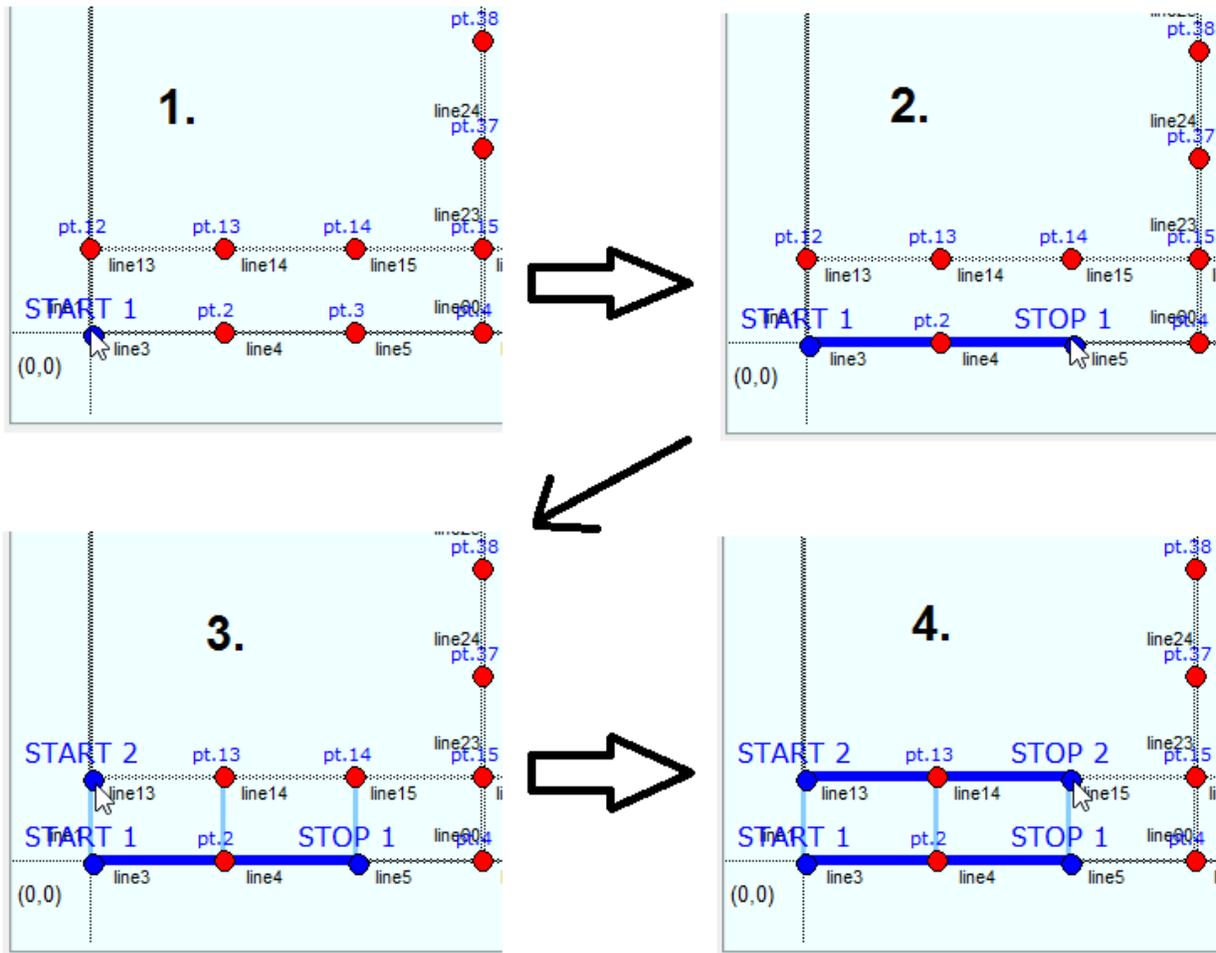
## Span optimization

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



The optimization is started as follows.

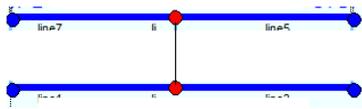
1. Start by clicking as illustrated next:



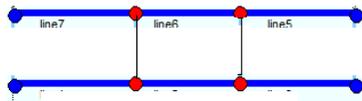
## 2. Then run optimization.

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.

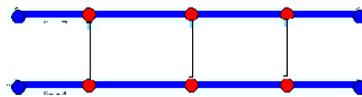
### 4 plate fields



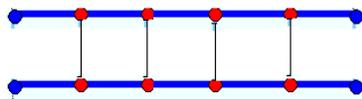
### 6 plate fields



### 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"/>	<b>Frame (girder data) for weight calculation:</b>	
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!

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?
*****			
<b>4</b>	<b>6.0</b>	<b>1.0</b>	<b>True</b>
<b>6</b>	<b>4.0</b>	<b>0.768</b>	<b>True</b>
<b>8</b>	<b>3.0</b>	<b>0.765</b>	<b>True</b>
<b>10</b>	<b>2.4</b>	<b>0.825</b>	<b>True</b>

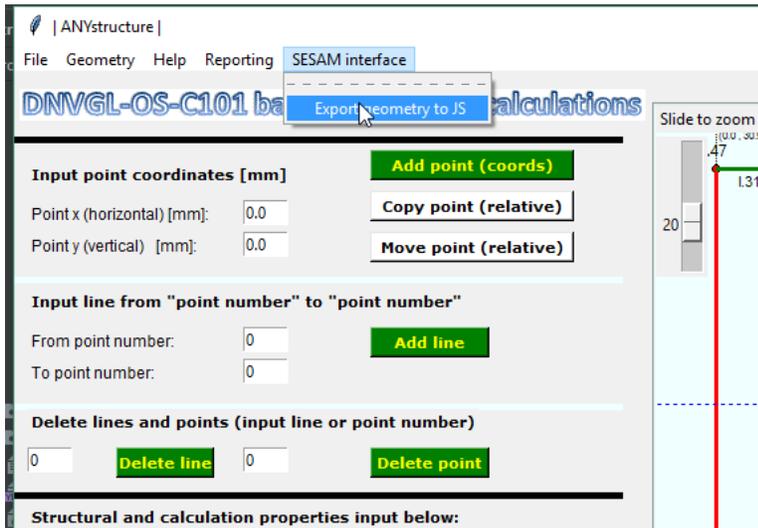
In this case 8 plate fields with length of 3 meter will give the lowest weight. 6 plate fields is almost equal.





# 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:

