



# ANYstructure

## Documentation

File Geometry Reporting SESAM interface Help

Input point coordinates [mm] **Add point (coords)**  
 Point x (horizontal) [mm]: 0.0 **Copy point (relative)**  
 Point y (vertical) [mm]: 0.0 **Move point (relative)**

Input line from "point number" to "point number"  
 From point number: 0 **Add line**  
 To point number: 0

Delete lines and points (or left/right click and use "Delete key")  
 Line number (left click): 8 **Delete line** **Delete prop.**  
 Point number (right click): 0 **Delete point**

Structural and calculation properties input below:

span	s	pL_thk	web_h	web_thk	fl_w	fl_thk
[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	stf type		
100.0	100.0	101.5	5.0	T		

FLS Material yield [MPa]: 355.0  
 Pressure side (p-plate, s-stf): p

Select structure type: **BOTTOM** (Internal, pressure from comp.)  
**Show structure types** **Add structure to line**

Find compartments External pressures

Comp. no.: 2, 3, 4, 5  
 Tank content: **Display current compartments**  
 Tank density: 0 [kg/m<sup>3</sup>] **Set compartment properties.**  
 Overpressure: 25000 [Pa] **Delete all tanks**  
 Max elevation: 0.0  
 Min elevation: 0.0  
 Acceleration [m/s<sup>2</sup>]:

Static and dynamic accelerations **line8**  
 Static acceleration [m/s<sup>2</sup>]: 9.81  
 Dyn. acc. loaded [m/s<sup>2</sup>]: 3.0 **Set accelerations**  
 Dyn. acc. ballast [m/s<sup>2</sup>]: 3.0

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

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

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]: [414334, 236275] DNV b [Pa]: [456615, 225481]  
 TT [Pa]: [335707] Manual [Pa]: [0,0]  
**Load factors** **Load info**

# 2021 Version 2.X

# 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>
Changing load factors	10
<b>Reviewing loads</b>	<b>11</b>
<b>Results</b>	<b>13</b>
<b>Optimization</b>	<b>13</b>
Optimization iteration by predefined stiffeners	13
Single optimization	14
Multiple optimization	17
Span optimization	18
<b>Reporting</b>	<b>23</b>
<b>Export to JS</b>	<b>24</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:

<b>CTRL-Z</b>	<b>Undo modelling</b>
<b>CTRL-C</b>	<b>Copy a selected point</b>
<b>CTRL-M</b>	<b>Move a selected point</b>
<b>CTRL-Q</b>	<b>New line between two selected points</b>
<b>CTRL-S</b>	<b>Assign properties to a selected line</b>
<b>CTRL-DELETE</b>	<b>Delete the structural properties from the selected line</b>

- DELETE**      Delete selected line/point
- CTRL-E**      Select a line and copy the properties of this line
- CTRL-D**      Paste structural 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. 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 displays a software window titled "Define structure properties". The interface is divided into several sections:

- Input fields:** "Input line from 'point number' to 'point number'", "From point number:" (0), "To point number:" (4), and "Delete lines and points" (4).
- Structural and calculation properties input below:** A table with columns for span, plate thickness, web height, web thickness, flange width, flange thickness, k<sub>pp</sub>, k<sub>ps</sub>, k<sub>m1</sub>, k<sub>m2</sub>, k<sub>3</sub>, sig<sub>y1</sub>, sig<sub>y2</sub>, sig<sub>x</sub>, tau<sub>y1</sub>, stf type, and pressure side. Values include 700.0, 18.0, 400.0, 12.0, 250.0, 14.0, 1.0, 1.0, 12.0, 24.0, 12.0, 100.0, 100.0, 100.5, 5.0, T, and p.
- Material yield:** 355.0 MPa.
- Select structure type:** BOTTOM.
- Buttons:** "Add line", "Delete line", "Add structure to line", "Find compartments", "Display current compartments", and "Save and return structure".
- Existing sections:** A diagram showing three cross-sections of a beam with dimensions b<sub>1</sub>, h, t, and b. A red box labeled "Existing sections" points to this diagram.
- Summary:** Plate: 700.0x18.0, Web: 400.0x12.0, Flange: 250.0x14.0.
- 3D Model:** A 3D perspective view of a girder with stiffeners, labeled "GIRDER", "PLATE", and "STIFFENER". The "Girder length (L<sub>g</sub>)" is set to 10.

Red callout boxes with arrows point to specific features:

- "Define plate and stiffener properties." points to the "Add structure to line" button.
- "Define buckling calculation properties." points to the "Find compartments" button.
- "Define fatigue properties." points to the "Delete all tanks" button.

## Display properties

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

The screenshot shows a software interface for defining tank properties. At the top, a horizontal line is divided into segments labeled 'l.pt. 1' through 'l.pt. 12'. The segment between 'l.pt. 6' and 'l.pt. 7' is highlighted in red and circled, with a red arrow pointing to the text 'Line 7' below it. The interface also displays a list of properties for the selected line, a diagram of a stiffener, and applied compartments and loads.

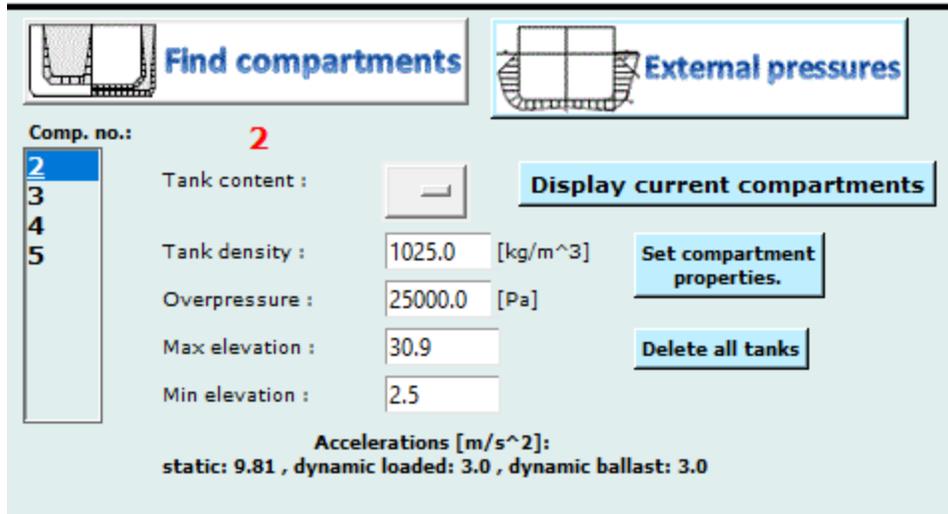
Plate field span:	3.6 meters	Applied compartments:	Compartment 4
Stiffener spacing:	750.0 mm		
Plate thickness:	18.0 mm		
Stiffener web height:	400.0 mm		
Stiffener web thickness:	12.0 mm		
Stiffener flange width:	250.0 mm		
Stiffener flange thickness:	12.0 mm		
Material yield:	355.0 MPa		
Structure 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		
km1,km2,km3:	12.0/24.0/12.0	Applied static/dynamic loads:	ballast_bottom loaded_static ballast_static loaded_bottom
Pressure side (p-plate/s-stf):	p		

## Define tanks

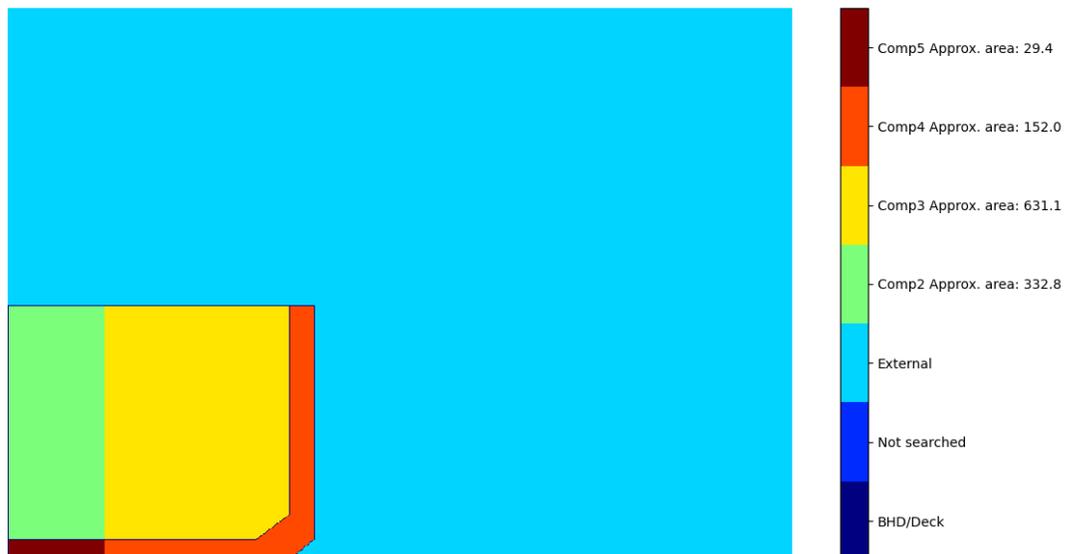
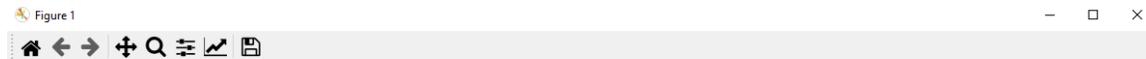
Tanks are searched for when clicking “Find compartments”. Non watertight structure are ignored. For information on structure types click “Show structure types”.

**By default tank content density is set to 0.**

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



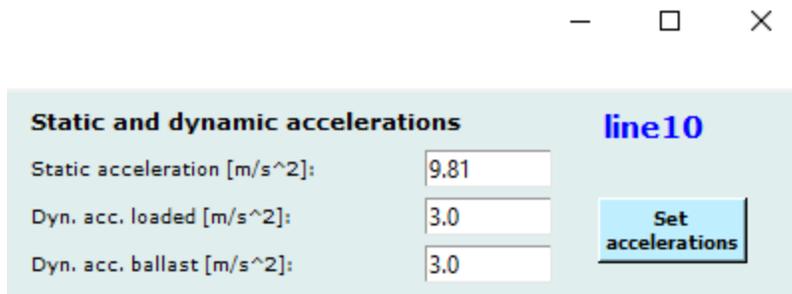
If you press “Display current compartments” after doing a compartment search, the result of the search is illustrated as seen next. Approximate area of the respective compartments is also shown.



\*area calculation inaccuracies due to thickness of barriers (BHD/Deck)

## Setting accelerations

Accelerations applies to tank content. I 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>]:

## 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.

The screenshot displays the 'Load properties' window with three main sections:

- 1. Dynamic loads:** Includes fields for 'Input load name' (ballast\_side), polynomial coefficients (Third, Second, First degree poly), 'Constant [C]', 'Load condition' (ballast), and 'Limit state' (ULS). A 'Create dynamic load' button is present.
- 2. Static loads:** Includes fields for 'Define name of static load' (static0), 'Define static draft from sea' (0.0), and 'Select load condition'. A 'Create static load' button is present.
- 3. Slamming pressure:** Includes fields for 'Load name' (slamming) and 'Pressure [Pa]' (0.0). A 'Create slamming load' button is present.

Below these sections is a grid for selecting lines to apply the load. The grid shows various line numbers (line1 to line57) and node numbers (ne1 to ne73). A vertical column of lines (Line 50 to Line 55) is highlighted in red. A 'Press to add selected lines to selected load' button is located above the grid. A 'Delete selected load' button is located in the top right of the grid area. A legend on the right side of the grid explains mouse and keyboard actions for selecting and clearing lines.

At the bottom left, a 'Properties selected load is:' box shows the configuration for the 'ballast\_side' load:

```

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.

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]

## Changing load factors

You can change default load factors and existing load factors using the button seen in the next illustration.

Load factors are based on standard DNV LRFD factors, but any values can be used.

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

name:	Stat LF	Dyn LF	Include?
static_22m	1.3	0	<input checked="" type="checkbox"/>
static_15m	1.3	0	<input checked="" type="checkbox"/>
static_8m_tt	0	0	<input type="checkbox"/>
loaded_bottom	0	0.7	<input checked="" type="checkbox"/>
ballast_bottom	0	0.7	<input checked="" type="checkbox"/>
Compartment2	1.2	0.7	<input checked="" type="checkbox"/>
Manual (pressure/LF)	0	1	<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]: [329265, 229422]      DNV b [Pa]: [298631, 212755]  
 TT [Pa]: [266326]      Manual [Pa]: [0,0]

**Load factors**      **Load info**

Load factor modifications here.

### Static and dynamic load factors is specified here

Note that DNV is used as reference, but the load factors can be any other rule set such as ISO.

Condition a) - Static load factor "unknown loads"   
 Condition a) - Static load factor well defined loads   
 Condition a) - Dynamic load factor

Condition b) - Static load factor "unknown loads"   
 Condition b) - Static load factor well defined loads   
 Condition b) - Dynamic load factor

Tank test) - Static load factor "unknown loads"   
 Tank test) - Static load factor well defined loads   
 Tank test) - Dynamic load factor

**Return specified load factors and change existing**

**Table 1 Load factors  $\gamma_f$  for ULS**

Combination of design loads	Load categories			
	G	Q	E	D
a)	1.3	1.3	0.7	1.0
b)	1.0	1.0	1.3	1.0

Load categories are:  
 G = permanent load  
 Q = variable functional load  
 E = environmental load  
 D = deformation load  
 For description of load categories see Sec.2.

4.4.2 When permanent loads (G) and variable functional loads (Q) are well defined, e.g. hydrostatic pressure, a load factor of 1.2 may be used in combination a) for these load categories.

4.4.3 If a load factor  $\gamma_f = 1.0$  on G and Q loads in combination a) results in higher design load effect, the load factor of 1.0 shall be used.

4.4.4 Based on a safety assessment considering the risk for both human life and the environment, the load factor  $\gamma_f$  for environmental loads may be reduced to 1.15 in combination b) if the structure is unmanned during extreme environmental conditions.

## Reviewing loads

Load calculations and results can be reviewed by clicking the "Load info" button. An example is seen in the next illustration.



## Load info for line9



Loads for condition: loaded - dnva  
 static with acceleration: 9.81 is:  
 $1 * 1.3 * 221215.5 = 287580.2$   
 dynamic with acceleration: 3.0 is:  
 $1 * 0.7 * 198687.0 = 139080.9$

RESULT:  $287580.2 + 139081 = 426661.1$

-----  
 Loads for condition: ballast - dnva  
 dynamic with acceleration: 3.0 is:  
 $1 * 0.7 * 62231.0 = 43561.7$   
 static with acceleration: 9.81 is:  
 $1 * 1.3 * 150828.8 = 196077.4$

comp4 - static:  $1 * 1.2 * 310707.225000000003 + 25000.0 * 1.3 = 405348.670000000004$   
 comp4 - dynamic:  $1 * 0.7 * 95017.500000000001 + 25000.0 * 0 = 66512.25$

RESULT:  $43561.7 + 196077 = 239639.0$

-----  
 Loads for condition: loaded - dnvb  
 static with acceleration: 9.81 is:  
 $1 * 1.0 * 221215.5 = 221215.5$   
 dynamic with acceleration: 3.0 is:  
 $1 * 1.3 * 198687.0 = 258293.2$

RESULT:  $221215.5 + 258293 = 479508.7$

-----  
 Loads for condition: ballast - dnvb  
 dynamic with acceleration: 3.0 is:  
 $1 * 1.3 * 62231.0 = 80900.2$   
 static with acceleration: 9.81 is:  
 $1 * 1.0 * 150828.8 = 150828.8$

comp4 - static:  $1 * 1.0 * 310707.225000000003 + 25000.0 * 1.3 = 343207.225000000003$   
 comp4 - dynamic:  $1 * 1.3 * 95017.500000000001 + 25000.0 * 0 = 123522.750000000003$

RESULT:  $80900.2 + 150829 = 231729.0$

-----  
 Tank test for: comp4  
 $1 * 1.0 * 310707.2 + 25000.0 * 1 = 335707$   
 Tank test for: comp4  
 $1 * 1.0 * 310707.2 + 25000.0 * 1 = 335707$   
 Manual pressure:  
 $0.0 * 1.0 * 1 = 0.0$

OK

## 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

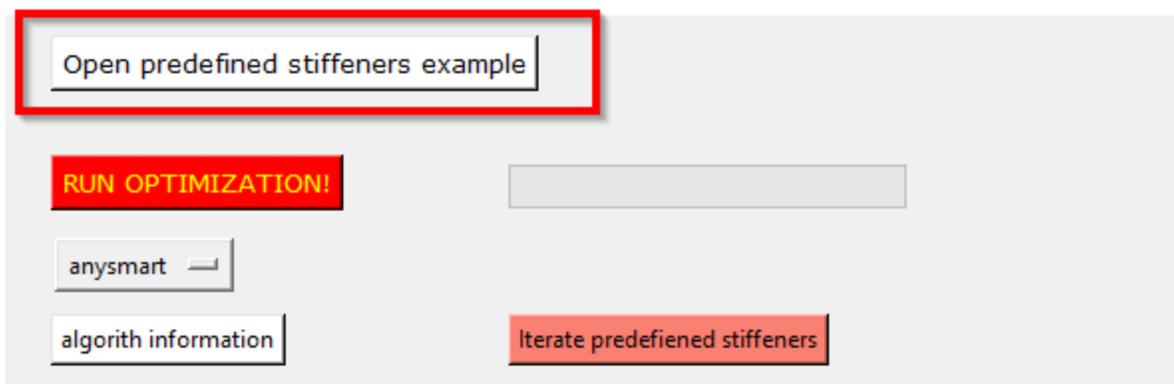
Section modulus: Wey1: 4.2400E+06 [mm<sup>3</sup>], Wey2: 1.4700E+06 [mm<sup>3</sup>]  
 Minimum section modulus: 2.0739E+06 [mm<sup>3</sup>]  
 Shear area: 4.6560E+03 [mm<sup>2</sup>]  
 Minimum shear area: 4.1297E+03 [mm<sup>2</sup>]  
 Plate thickness: 18.0 [mm]  
 Minimum plate thickness: 15.8 [mm]  
 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.

**-- 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]	<input type="text" value="850.0"/>	<input type="text" value="25.0"/>	<input type="text" value="600.0"/>	<input type="text" value="35.0"/>	<input type="text" value="300.0"/>	<input type="text" value="40.0"/>
Iteration delta [mm]	<input type="text" value="50.0"/>	<input type="text" value="2.0"/>	<input type="text" value="50.0"/>	<input type="text" value="2.0"/>	<input type="text" value="50.0"/>	<input type="text" value="2.0"/>
Lower bounds [mm]	<input type="text" value="650.0"/>	<input type="text" value="10.0"/>	<input type="text" value="400.0"/>	<input type="text" value="15.0"/>	<input type="text" value="100.0"/>	<input type="text" value="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

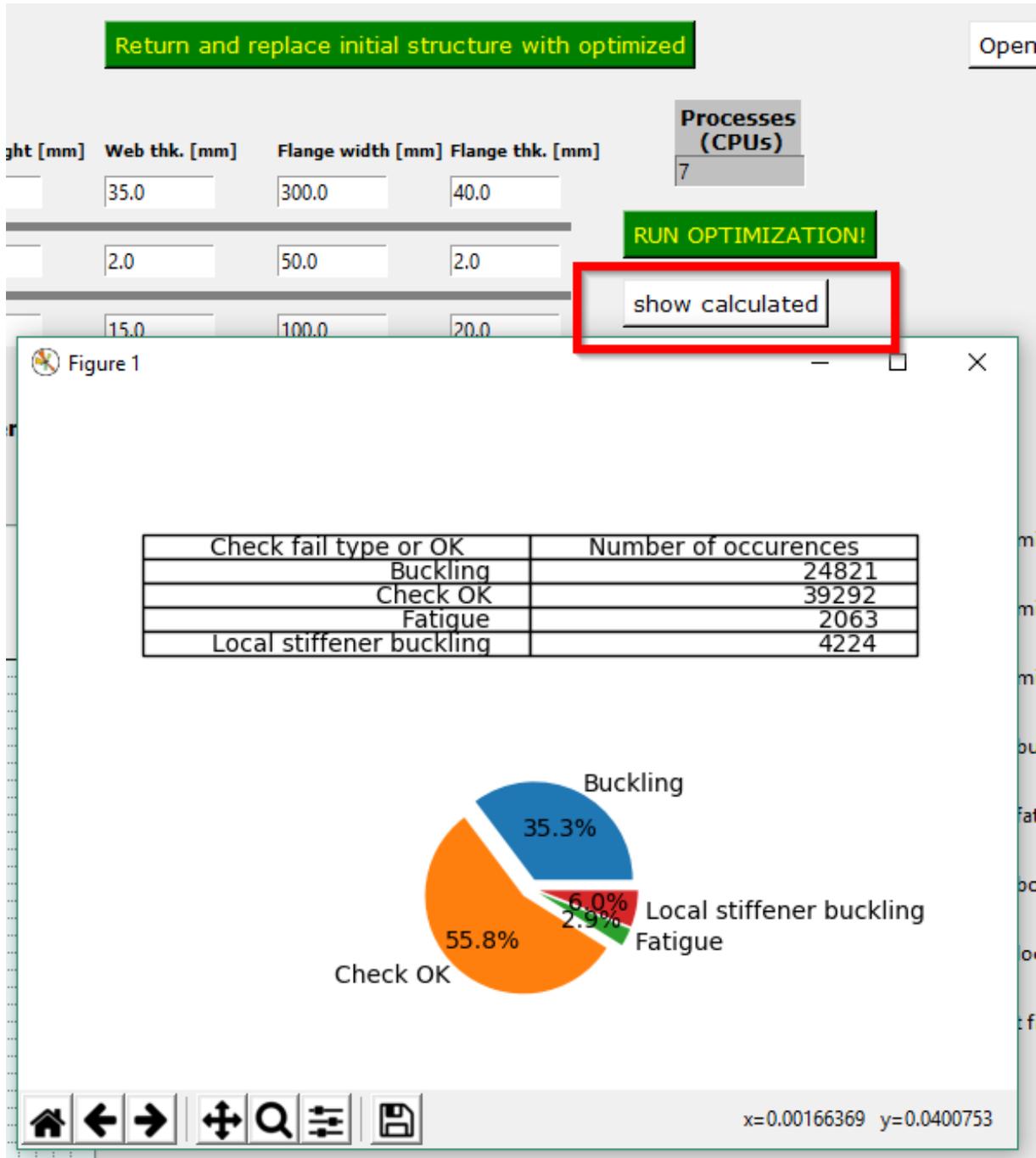
Various checks in the optimization module:

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

The screenshot displays the 'Optimize structure' software interface. At the top, it shows the title 'Optimize structure' and a window title bar. The main area is titled '-- Structural optimizer for multiple selections --'. It features a table for input parameters:

Upper bounds [mm]	Spacing [mm]	Plate thk. [mm]	Web height [mm]	Web thk. [mm]	Flange width [mm]	Flange thk. [mm]
800	25	500	22	250	30	
Iteration delta [mm]	50.0	2.0	50.0	2.0	50.0	2.0
Lower bounds [mm]	600	10	300	10	50	10

Below the table, it shows 'Estimated running time for algorithm: 0 seconds'. To the right, there are buttons for 'RUN OPTIMIZATION!', 'Return and replace with selected optimized structure', and 'Iterate predefined stiffeners'. A 'Processes (CPUs)' section shows '11' CPUs. A checkbox is checked for 'Check to harmonize results. Same stiffener and plate dimensions. (defined by largest in opt.)'. A structural diagram on the left shows a rectangular plate with stiffeners labeled 'line1' through 'line13'. A legend on the right provides optimization results:

- Initial - Pl: 750.0x15.0 Stf: 400.0x18.0+150.0x20.0
- Weight (per Lg width): 8873
- Optimized - Pl: 600.0x14.0 Stf: 350.0x12.0+150.0x10.0
- Weight (per Lg width): 7359

Additional options include 'Check for minimum section modulus', 'Check for minimum plate thk.', 'Check for minimum shear area', 'Check for buckling (RP-C201)', 'Check for fatigue (RP-C203)', 'Check for bow slamming', and 'Check for local stf. buckling'.

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. Remember to select the lines you want to return.

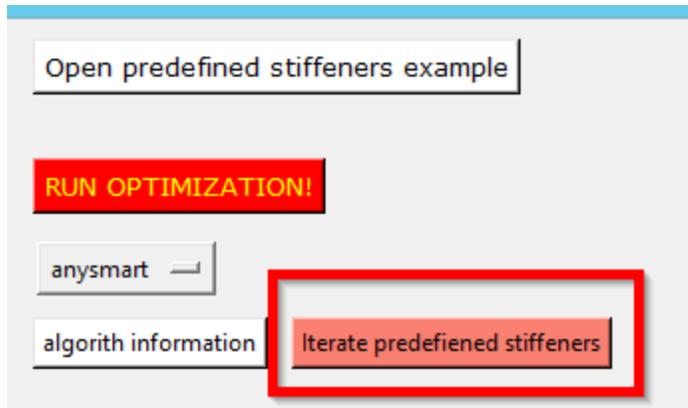
The optimization can be **harmonized**. That means that the largest dimension found in the multiple optimization is used for all selected. This is done after all plates/stiffeners are checked. Harmonization can only be done in the multipt option

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

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

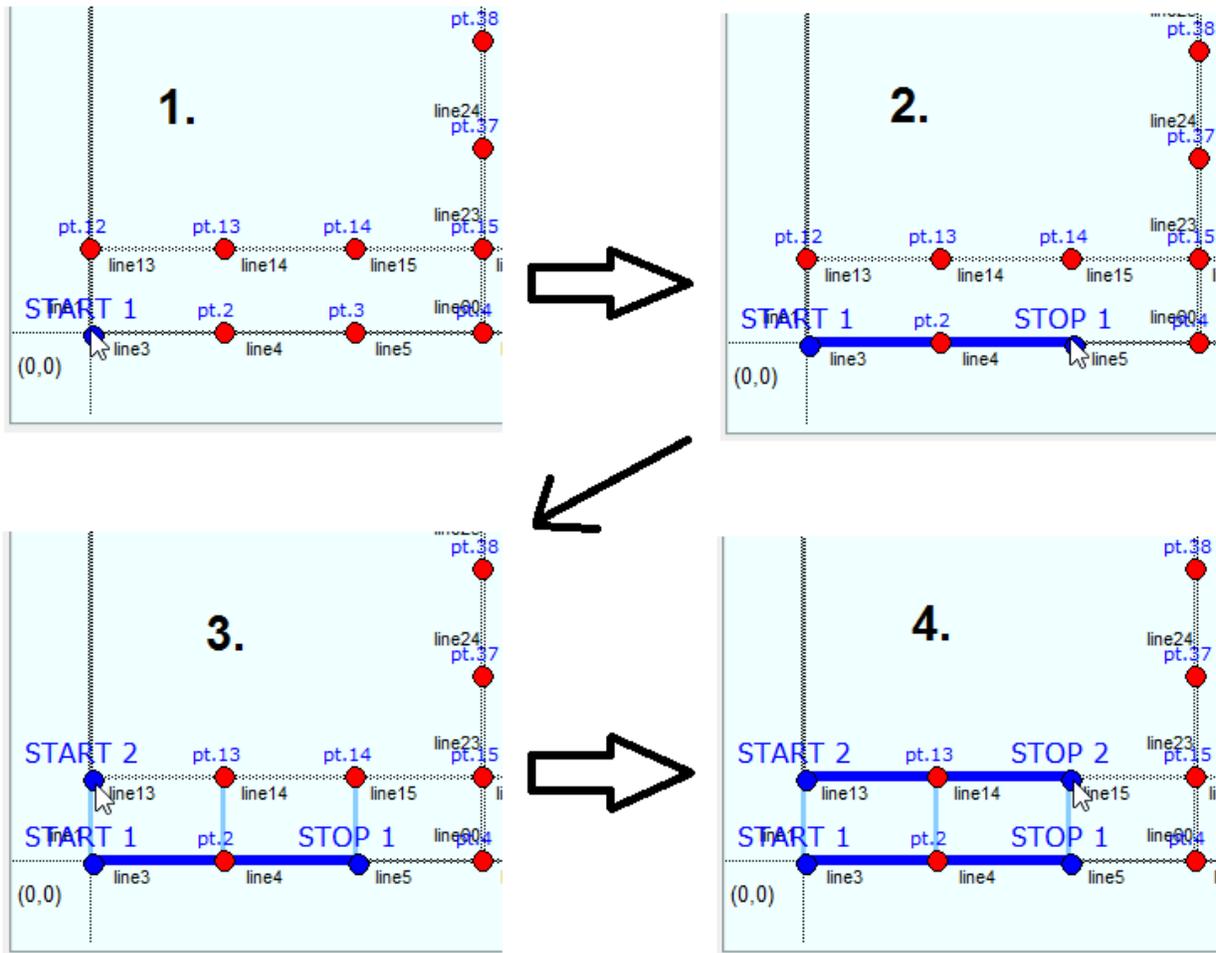
## Span optimization

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



The optimization is started as follows.

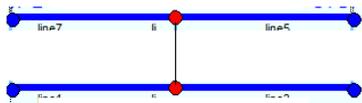
1. Start by clicking as illustrated next:



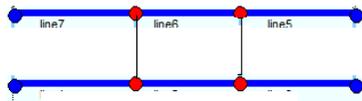
## 2. Then run optimization.

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.

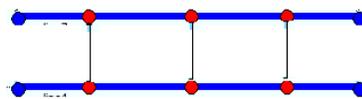
### 4 plate fields



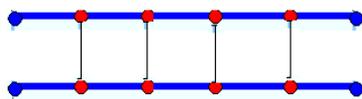
### 6 plate fields



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

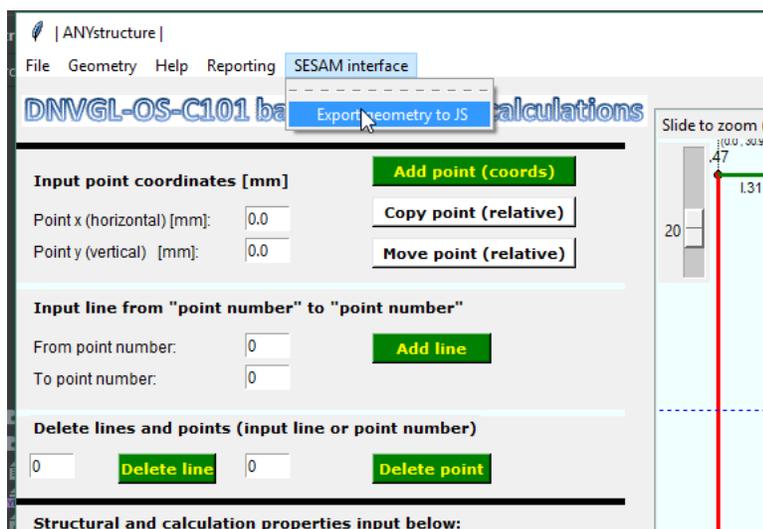
In this case 8 plate fields with length of 3 meter will give the lowest weight. 6 plate fields is almost equal.





## 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:

