



# ANYstructure

File Geometry Reporting SESAM interface Help

Input point coordinates [mm] **Add point (coords)**  
 Copy point (relative)  
 Move point (relative)

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

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): 7 **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
3.6	750.0	18.0	400.0	12.0	250.0	12.0
[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	stf type		
100.0	100.0	101.5	5.0	T		

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

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

Find compartments External pressures  
 Tank content: 0 [kg/m³]  
 Tank density: 0 [kg/m³]  
 Overpressure: 25000 [Pa]  
 Max elevation: 0.0  
 Min elevation: 0.0  
 Acceleration [m/s²]: 0.0

**Display current compartments**  
**Set compartment properties.**  
**Delete all tanks**

Check to see available shortcuts

CTRL-Z Undo geometry action  
 CTRL-C Copy selected points  
 CTRL-M Move selected points  
 CTRL-Q New line (right click two points)  
 CTRL-S Assign structure properties to clicked line  
 CTRL-DELETE Delete structure properties from clicked line  
 DELETE Delete active line and/or point  
 CTRL-E Copy line properties from active line  
 CTRL-D Paste line properties to active line  
 Mouse click: left/right - select line/point  
 Arrow left/right - previous/next line

Static and dynamic accelerations **line 7**  
 Static acceleration [m/s²]: 9.81  
 Dyn. acc. loaded [m/s²]: 3.0 **Set accelerations**  
 Dyn. acc. ballast [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 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 th, 4.3.7 and 4.3.8 is accounted for.  
 DNV a [Pa]: {453279, 233557} DNV b [Pa]: {439799, 220435}  
 TT [Pa]: {335707} Manual [Pa]: {0,0}

**Load factors** **Load info**

Plate field span: 3.6 meters  
 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  
 Dynamic load variable: x - horizontal  
 Plate fixation parameter:kpp: 1.0  
 Global stress: sig\_y1/sig\_y2: 100.0/100.0 MPa  
 Global stress: sig\_x: 101.5 MPa  
 Global shear: tau\_y1: 5.0 MPa  
 Int.Lim2.km2: 12.0/24.0/12.0  
 Pressure side (p-plate/s-stf): p

SELECTED: line 7

Applied compartments: Compartment 4

Applied static/dynamic loads:  
 ballast\_bottom  
 loaded\_static  
 ballast\_static  
 loaded\_bottom

Section modulus: Wey1: 4.5300E+06 [mm³], Wey2: 1.7500E+06 [mm³]  
 Minimum section modulus: 1.7163E+06 [mm³]

Shear area: 5.1608E+03 [mm²]  
 Minimum shear area: 3.5276E+03 [mm²]

Plate thickness: 18.0 [mm]  
 Minimum plate thickness: 15.1 [mm]

Buckling results DNV-RP-C201:  
 Eq 7.19: 0.68 [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

# Documentation

## 2021 Version 2.X

# Table of contents

<b>Table of contents</b>	<b>2</b>
<b>Introduction</b>	<b>3</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>7</b>
<b>Setting accelerations</b>	<b>8</b>
<b>Define external pressures</b>	<b>9</b>
<b>Load combinations</b>	<b>10</b>
Changing load factors	11
<b>Reviewing loads</b>	<b>12</b>
<b>Results</b>	<b>14</b>
<b>Optimization</b>	<b>14</b>
Optimization iteration by predefined stiffeners	14
Single optimization	15
Multiple optimization	18
Span optimization	19
<b>Reporting</b>	<b>24</b>
<b>Export to JS</b>	<b>25</b>

## Introduction

ANYstructure is a free structural optimization tool. It can be used for multiple purposes. The software can be downloaded various ways:

For python users

PIP install ANYstructure

For windows version

Download at <https://github.com/audunarn/ANYstructure/releases> or  
<https://sourceforge.net/projects/anystructure/>

The code is located on github and is open source (<https://github.com/audunarn/ANYstructure>)

## 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-P</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>

<b>DELETE</b>	<b>Delete selected line/point</b>
<b>CTRL-E</b>	<b>Select a line and copy the properties of this line</b>
<b>CTRL-D</b>	<b>Paste structural properties to a selected line</b>

**Left and right arrow to change current line.**

**Up and down arrow to change current 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”.**

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: Define plate and stiffener properties.

Delete lines and points

4 Delete line

Structural and calculation properties input below:

span	pl_thk	web_h	web_thk	fl_w	fl_thk
5.9	700.0	18.0	400.0	12.0	250.0
[m]	[mm]	[mm]	[mm]	[mm]	[mm]
kpp	kps	km1	km2	k3	
1.0	1.0	12.0	24.0	12.0	
[mm]	[mm]	[mm]	[mm]	[mm]	
sig_y1	sig_y2	sig_x	tau_y1	stf type	pressure side
100.0	100.0	100.5	5.0	IT	p
[MPa]	[MPa]	[MPa]	[MPa]		

Material yield (MPa) 355.0

Select structure type -> BOTTOM

Internal pressure from comp. Add structure to line

Find compartments

Display current compartments

Comp. no.:

2
3
4
5

Tank content:

Tank density: 1025 [kg/m<sup>3</sup>]

Overpressure: 25000 [Pa] Delete all tanks

Acceleration [m/s<sup>2</sup>]:

Define buckling calculation properties.

Define fatigue properties.

Define structure properties here --

type: T

700.0 [mm]

18.0 [mm]

400.0 [mm]

12.0 [mm]

250.0 [mm]

14.0 [mm]

Existing sections:

Plate: 700.0x18.0  
Web: 400.0x12.0  
Flange: 250.0x14.0

Girder length (Lg) 10

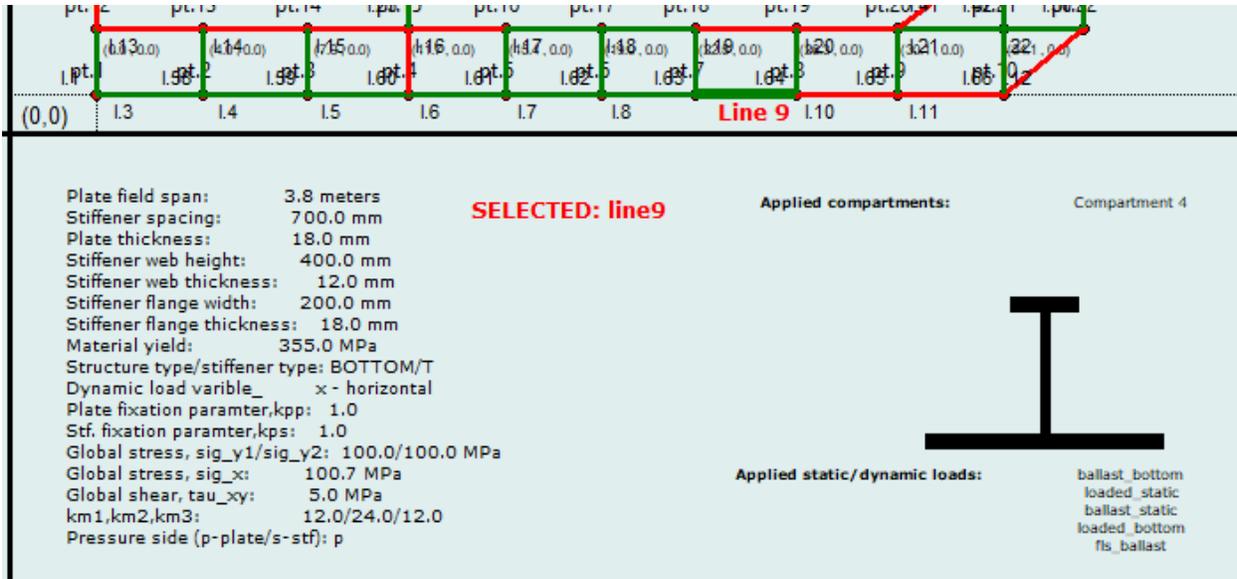
Save and return structure

By default  $z^*$  is ticked. This affects the buckling results and will generally give lower utilization than using  $z^* = 0$ . See description below.

$z^*$  is the distance from the neutral axis of the effective section to the working point of the axial force.  $z^*$  may be varied in order to optimise the resistance.  $z^*$  should then be selected so the maximum utilisation found from the equations (7.50) to (7.53) or (7.54) to (7.57) is at its minimum, see also Commentary Chapter 10. The value of  $z^*$  is taken positive towards the plate. The simplification  $z^* = 0$  is always allowed.

## Display properties

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

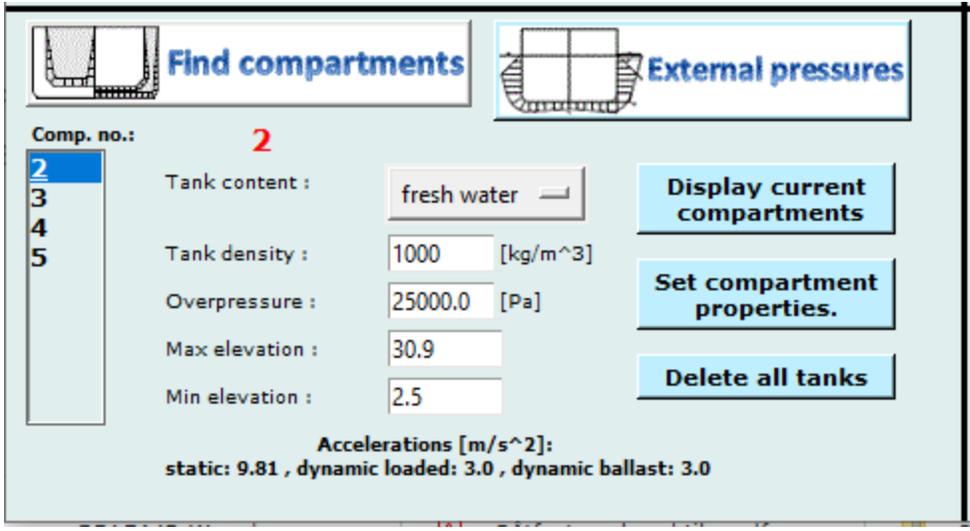


## Define tanks

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.

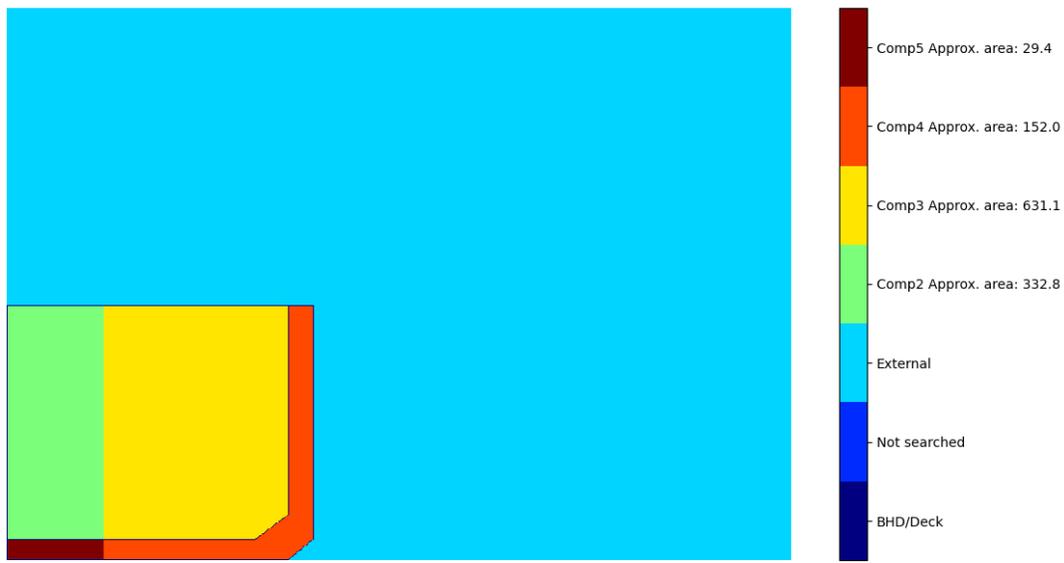
Ather tanks are found content and overpressure must be defined as seen next.



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.



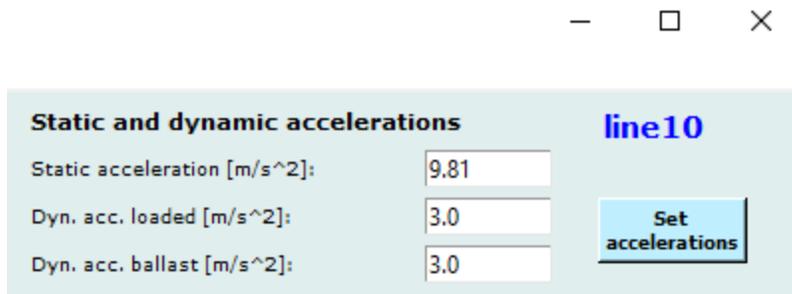
Compartments returned from search operation displayed below



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

## Setting accelerations

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



## 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

Z (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 line58) 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.

Instructions for the grid:
 

- 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

At the bottom left, a 'Properties selected load is:' box shows details for the 'ballast\_side' load, including its name, polynomial coefficients, constant, load condition, limit state, and draft.

## 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.

DS-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

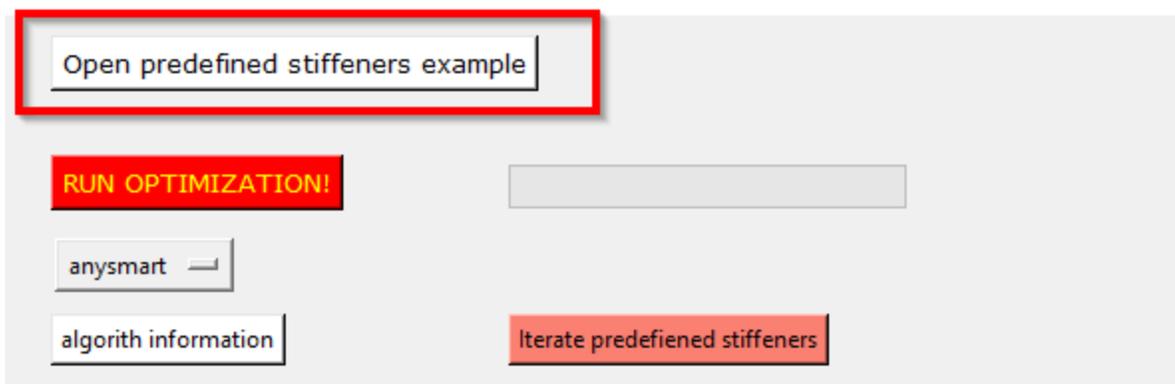
Section modulus: Wey1: 4.2400E+06 [mm<sup>3</sup>], Wey2: 1.4700E+06 [mm<sup>3</sup>]  
 Minimum section modulus: 2.0739E+06 [mm<sup>3</sup>]  
 Shear area: 4.6560E+03 [mm<sup>2</sup>]  
 Minimum shear area: 4.1297E+03 [mm<sup>2</sup>]  
 Plate thickness: 18.0 [mm]  
 Minimum plate thickness: 15.8 [mm]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

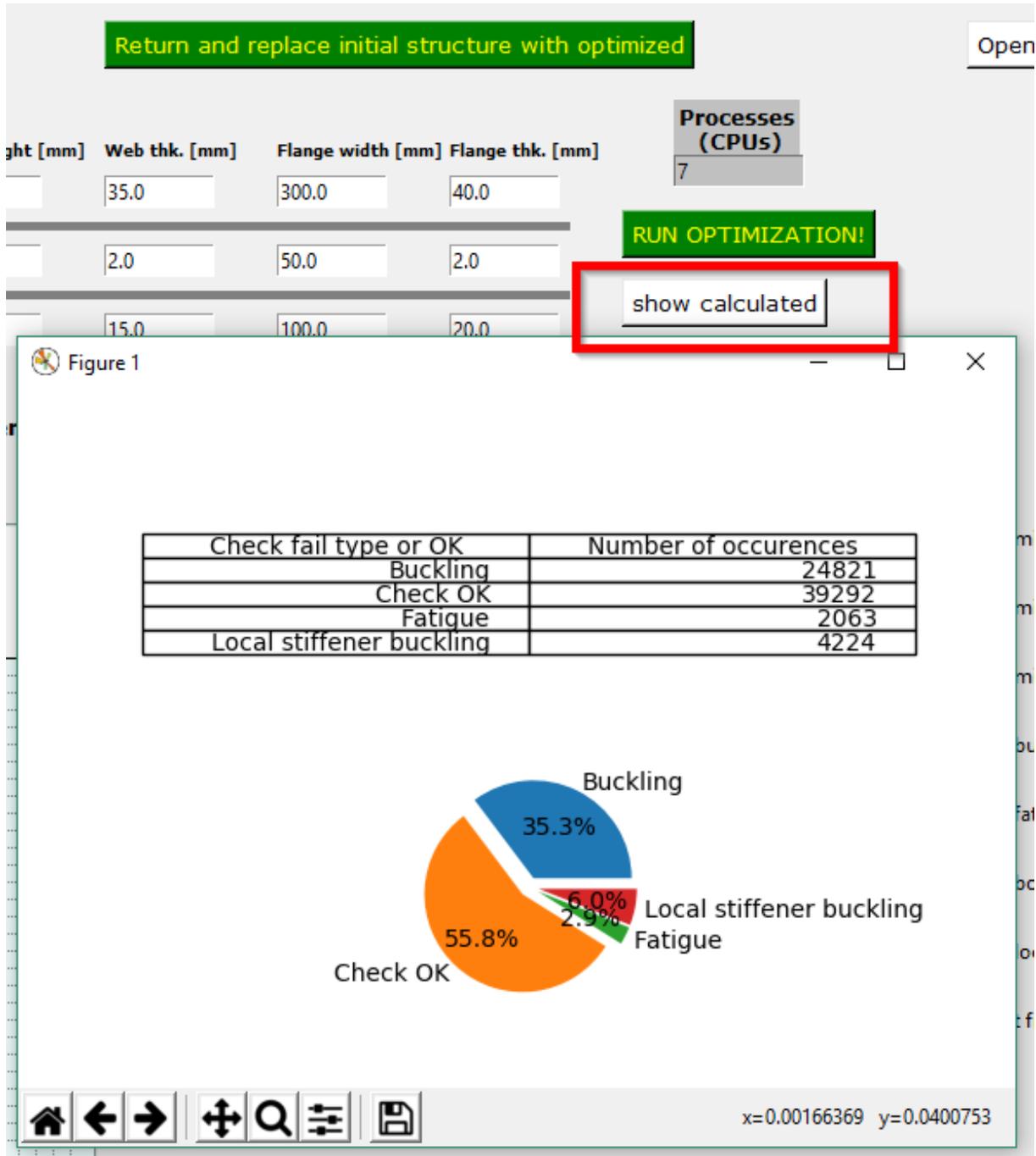
Various checks in the optimization module:

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

The screenshot shows the 'Optimize structure' software interface. At the top, it says '-- Structural optimizer for multiple selections --'. Below this are input fields for 'Upper bounds [mm]', 'Iteration delta [mm]', and 'Lower bounds [mm]' for various parameters like Spacing, Plate thk., Web height, Web thk., Flange width, and Flange thk. A 'RUN OPTIMIZATION' button is visible. To the right, there's a 'Return and replace with selected optimized structure' button and a 'Processes (CPUs)' section. A central diagram shows a plate with lines labeled line0 to line14. A right-hand panel displays optimization results, including 'Initial' and 'Optimized' plate specifications, weight per length, and a list of checked options like '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 slumping', and 'Check for local stf. buckling'.

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

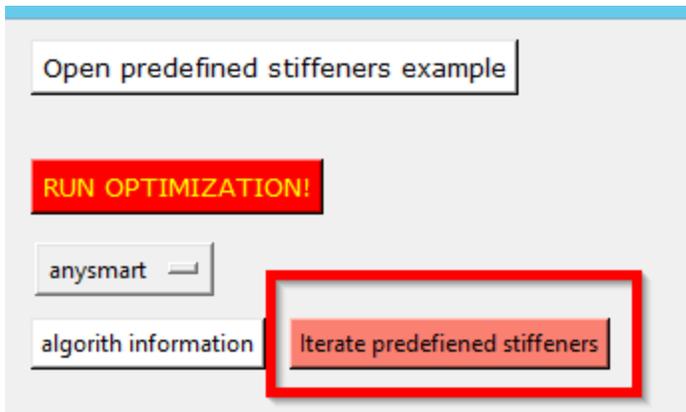
1. Same input on upper bounds, lower bounds and delta.
2. Click all the lines you want to include in the optimization.
3. Run the optimization.
4. Check the properties by **middle clicking** the line you ran.
5. If you are happy return the properties by clicking the top button. Remember to select the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

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 multiopt 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).

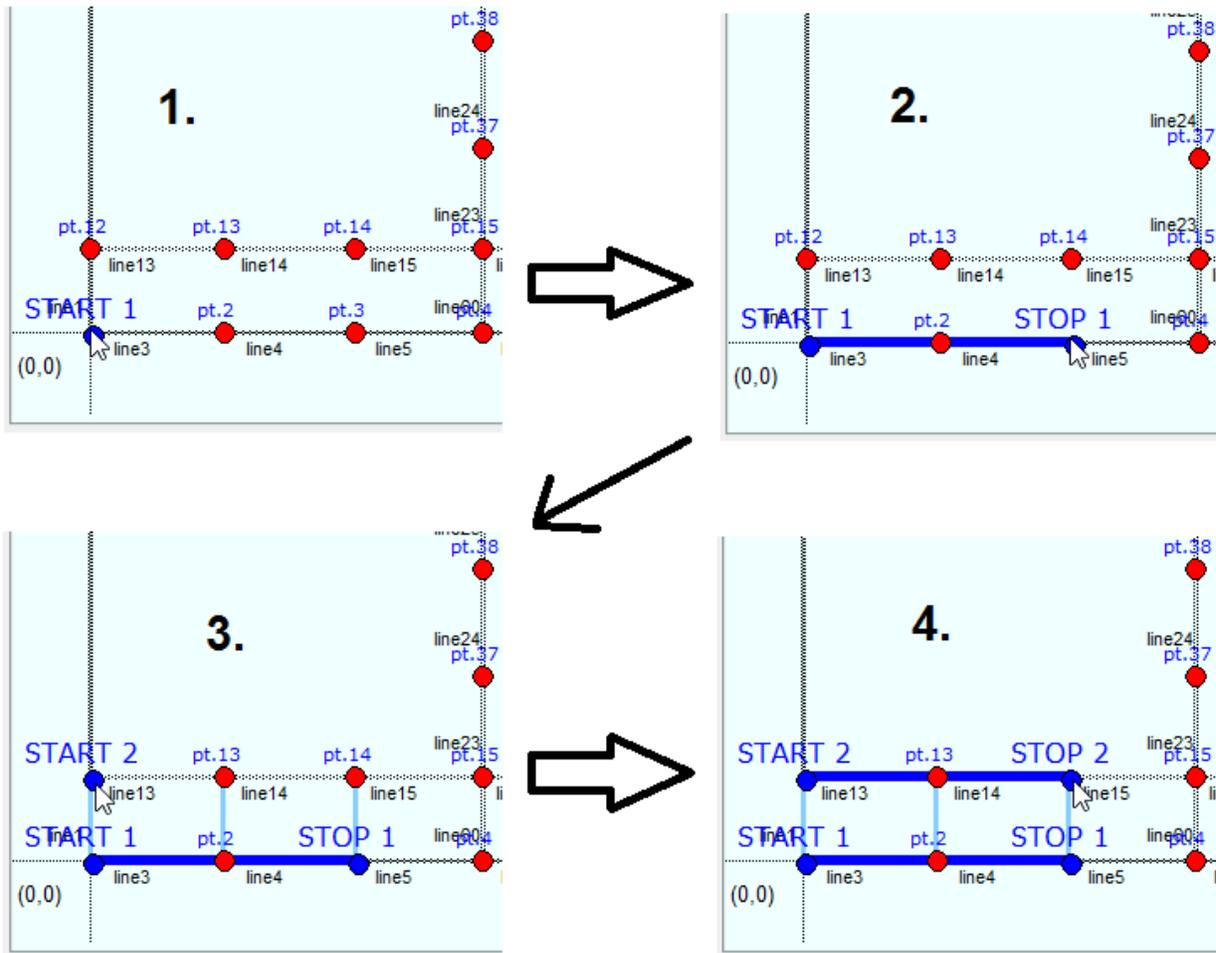
## Span optimization

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



The optimization is started as follows.

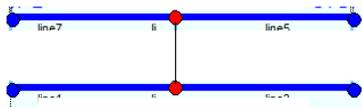
1. Start by clicking as illustrated next:



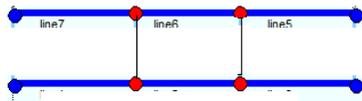
## 2. Then run optimization.

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.

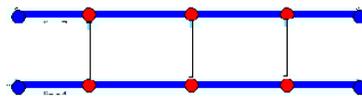
### 4 plate fields



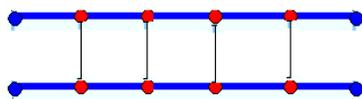
### 6 plate fields



### 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?
*****			
<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>

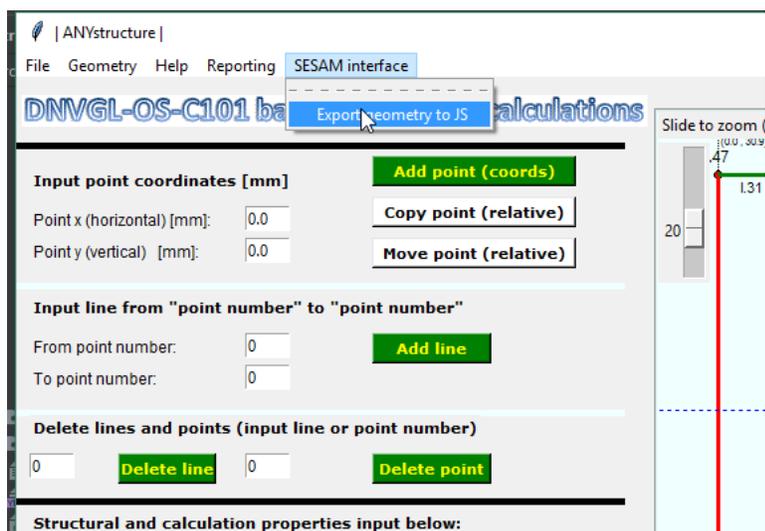
In this case 8 plate fields with length of 3 meter will give the lowest weight. 6 plate fields is almost equal.





## 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:

