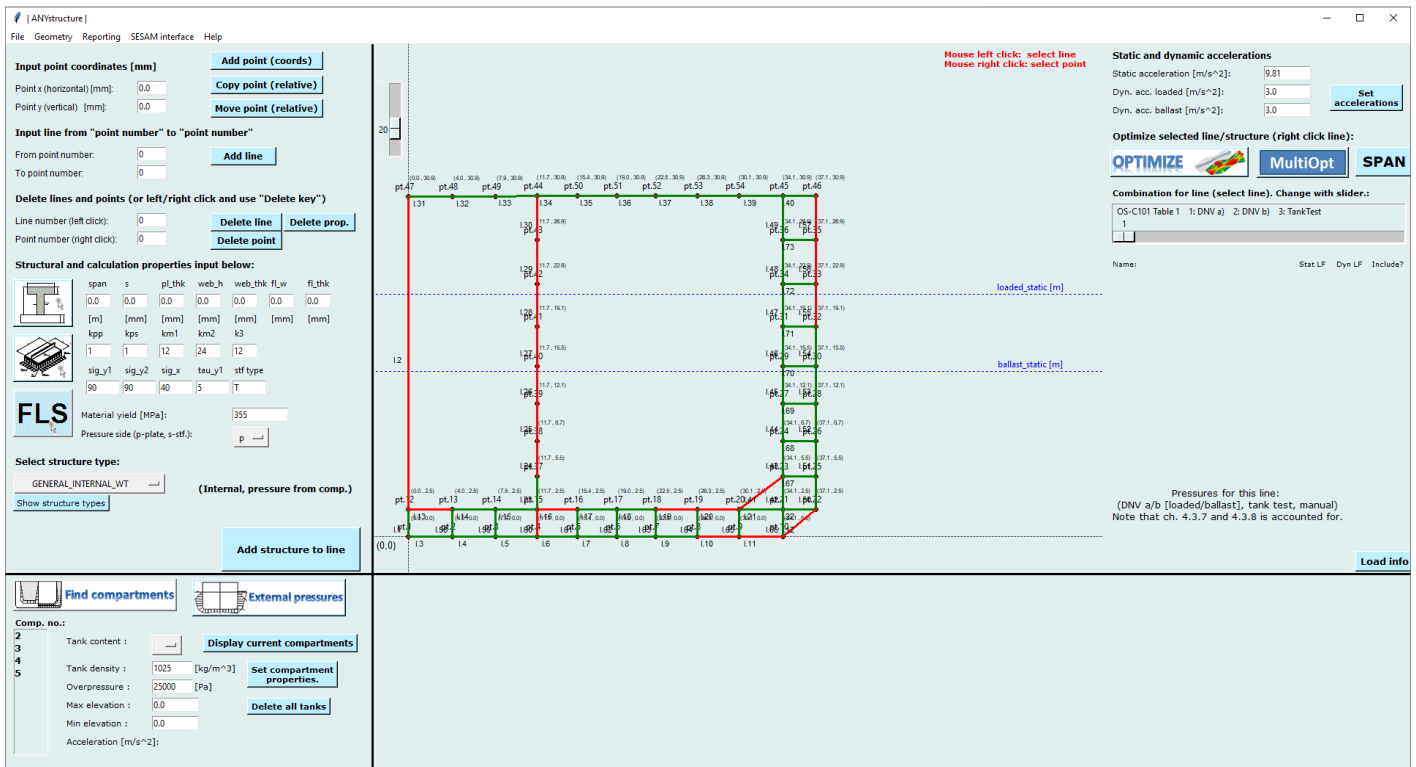




# ANYstructure

## Documentation



2020/2021

Version 1.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>
<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>

The logo for DNV-GL, consisting of the letters 'DNV-GL' in a bold, dark blue, sans-serif font.

# 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>	<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 shows the 'Define structure properties' window with several red callout boxes and arrows pointing to specific features:

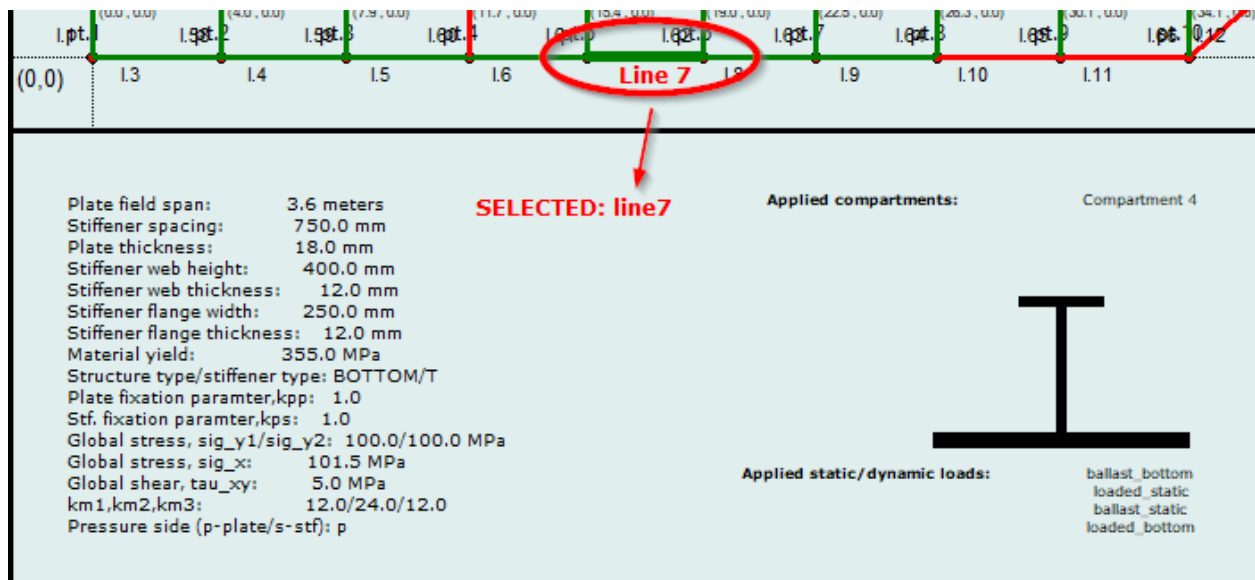
- Define plate and stiffener properties.** Points to the 'Add line' button in the 'Input line from "point number" to "point number"' section.
- Existing sections** points to the 'Existing sections' dropdown menu in the 'Define structure properties here --' section.
- Define buckling calculation properties.** Points to the 'Find compartments' button in the 'Structural and calculation properties input below:' section.
- Define fatigue properties.** Points to the 'Delete all tanks' button in the 'Comp. no.:' section.

The interface includes the following sections and data:

- Input line from "point number" to "point number":** From point number: 0, To point number: 4. Buttons: Add line, Delete line.
- Structural and calculation properties input below:**
  - span: 700.0 [mm]
  - pl\_thk: 18.0 [mm]
  - web\_h: 400.0 [mm]
  - web\_thk fl\_w: 12.0 [mm]
  - fl\_thk: 14.0 [mm]
  - kpp: 1.0 [mm]
  - kps: 1.0 [mm]
  - km1: 12.0 [mm]
  - km2: 24.0 [mm]
  - k3: 12.0 [mm]
  - sig\_y1: 100.0 [MPa]
  - sig\_y2: 100.0 [MPa]
  - sig\_x: 100.5 [MPa]
  - tau\_y1: 5.0 [MPa]
  - stf type: T
  - pressure side: p
  - Material yield [MPa]: 355.0
  - Select structure type -> BOTTOM
  - Buttons: Show structure types, Add structure to line
- Define structure properties here --**
  - type: T
  - Plate thk: 18.0 [mm]
  - Web height: 400.0 [mm]
  - Web thk: 12.0 [mm]
  - Flange width: 250.0 [mm]
  - Flange thk: 14.0 [mm]
  - Existing sections: (dropdown menu)
  - Summary: Plate: 700.0x18.0, Web: 400.0x12.0, Flange: 250.0x14.0
  - Diagram: A cross-section of a T-beam with dimensions b1, h, tw, tf, b, and z.
  - 3D Diagram: A 3D view of a girder with stiffeners, labeled with GIRDER, PLATE, and STIFFENER. Dimensions include Lg (Girder length) and l.
  - Buttons: Save and return structure
- Comp. no.:** 2, 3, 4, 5. Tank content: 1025 [kg/m³], Tank density: 75000 [Pa]. Acceleration [m/s²]:

## Display properties

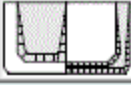
If you click a line properties is displayed in the window below as seen next.



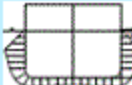
## Define tanks

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

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



### Find compartments



### External pressures

Comp. no.: 2

2

3

4

5

Tank content :

Display current compartments

Tank density :

[kg/m<sup>3</sup>]

Set compartment properties.

Overpressure :

[Pa]

Max elevation :

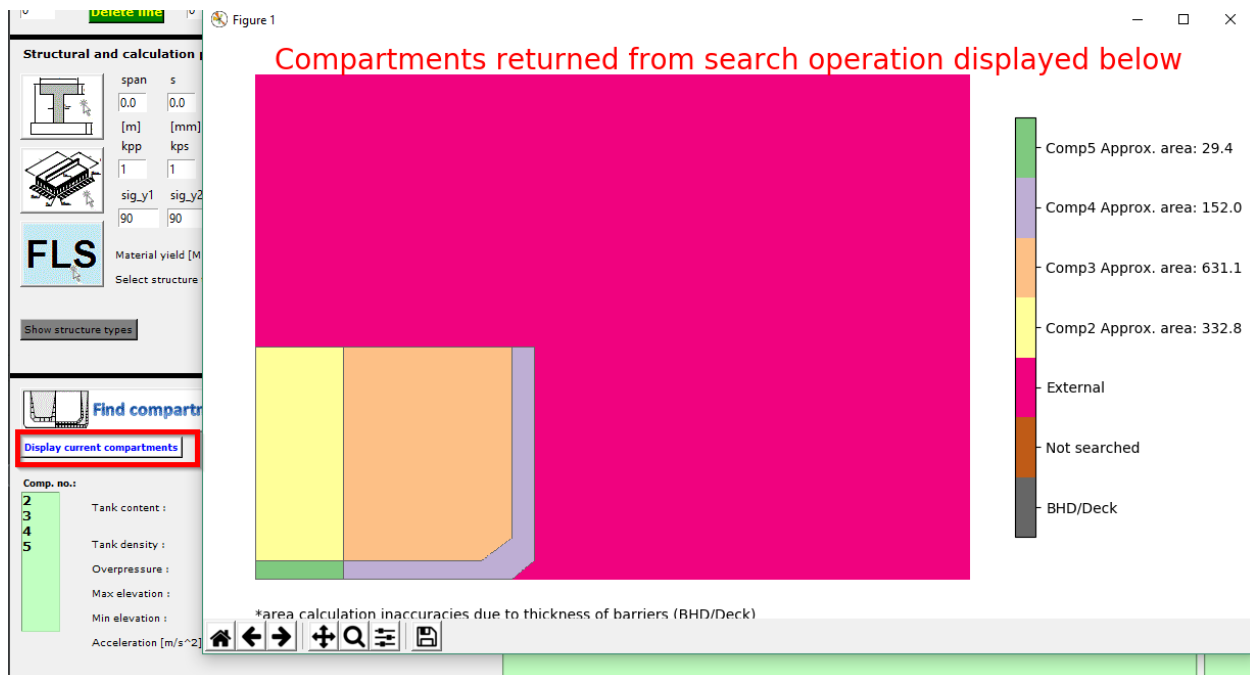
Min elevation :

Delete all tanks

**Accelerations [m/s<sup>2</sup>]:**

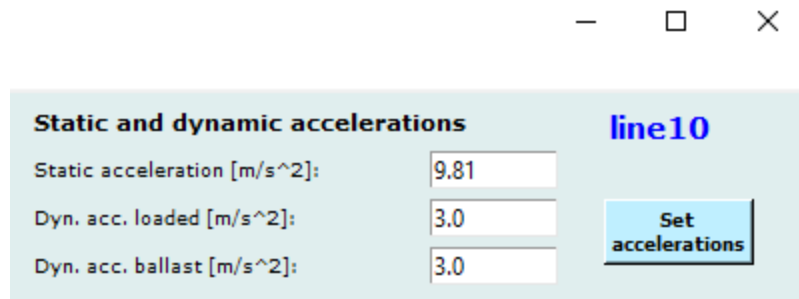
static: 9.81 , dynamic loaded: 3.0 , dynamic ballast: 3.0

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.



## 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**

**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.

Load properties

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:

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

line31 line32 line33 line34 line35 line36 line37 line38 line39 line40

line49 line57

line73

line48 line56

line72

line47 Line 55

line71

line46 Line 54

line70

line45 Line 53

line69

line44 Line 52

line68

line43 Line 51

line67

Line 50

line42

line21

line66

line22

line13 line14 line15 line16 line17 line18 line19 line20 line21 line22

line59 line60 line61 line62 line63 line64 line65 line66 line67 line68 line69 line70 line71 line72

line1 line2

(0,0)

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

ballast\_bottom

loaded\_static

ballast\_static

slamming

loaded\_bottom

fls\_ballast

line50

line51

line52

line53

line54

line55

Properties selected load is:

Name of load: ballast\_side

Polynomial (x^3): 0.0

Polynomial (x^2): 303.0

Polynomial (x): -3750.0

Constant (C): 153000.0

Load condition: ballast

Limit state: ULS

Is external?: True

Static draft: None

## Load combinations

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

- 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.

Open predefined stiffeners example

RUN OPTIMIZATION!

anysmart

algorithm information

Iterate predefined stiffeners

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

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

-- Structural optimizer --

Return and replace initial structure with optimized

Iterate predefined stiffeners

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

Estimated running time for algorithm: 7 seconds

RUN OPTIMIZATION!

## Single optimization

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

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

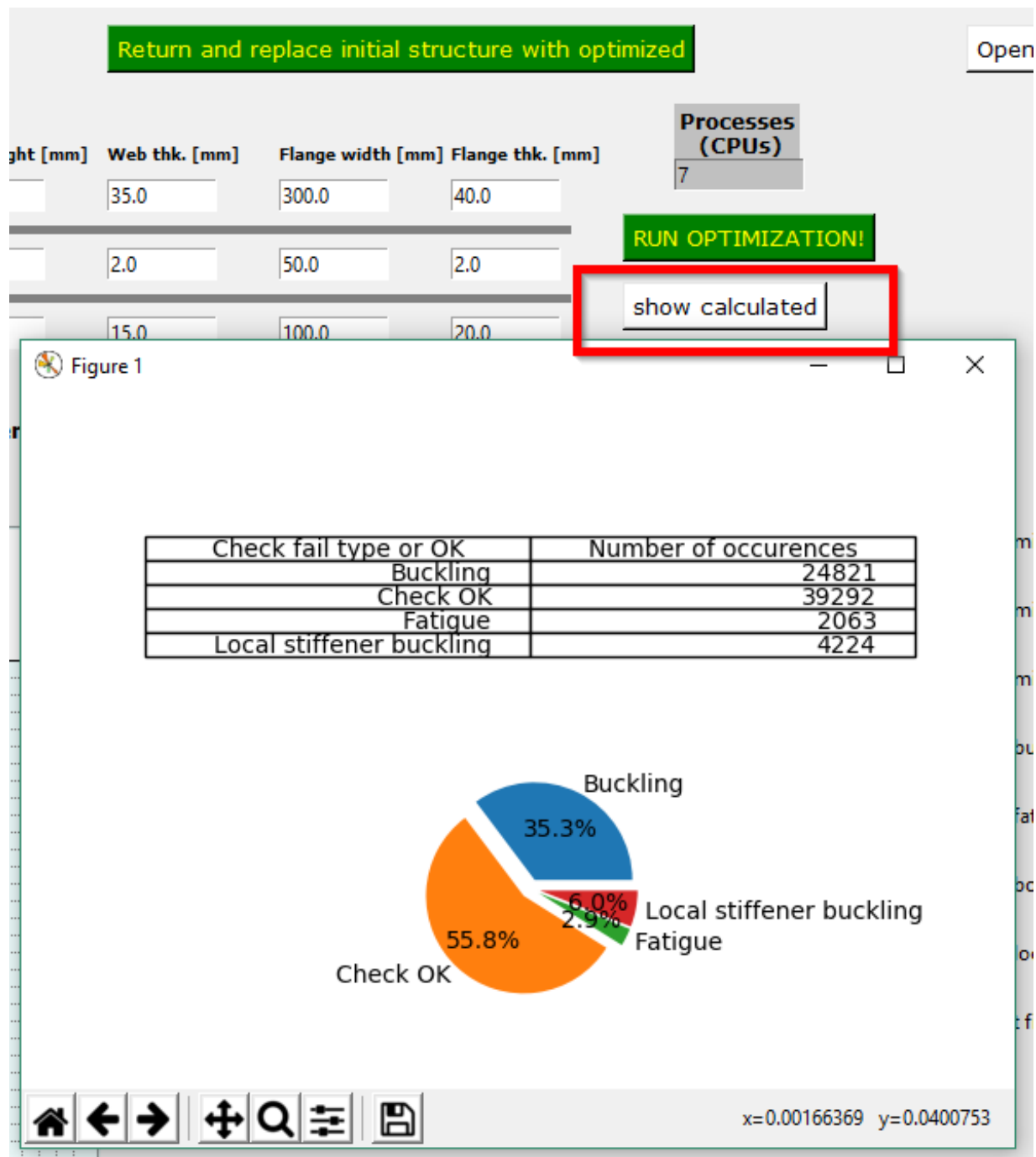
Various checks in the optimization module:

You can select the checks to be performed.

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

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

If you press the “show calculated” button, you will get an overview of how many is ok and how many failed (and what criteria first failed). One “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.

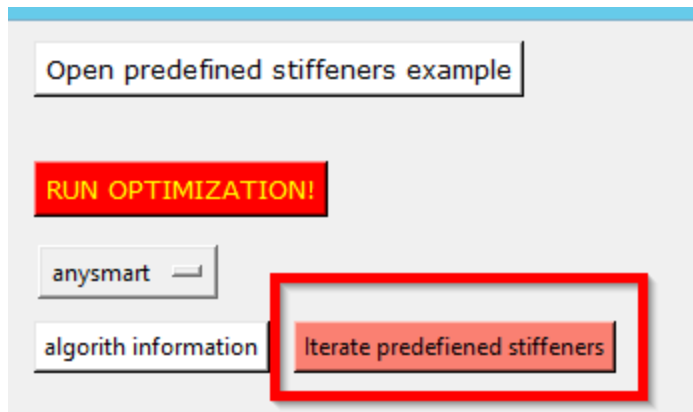
1. Same input on upper bounds, lower bounds and delta.
2. Click all the lines you want to include in the optimization.
3. Run the optimization.
4. Check the properties by **middle clicking** the line you ran.
5. If you are happy return the properties by clicking the top button

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

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

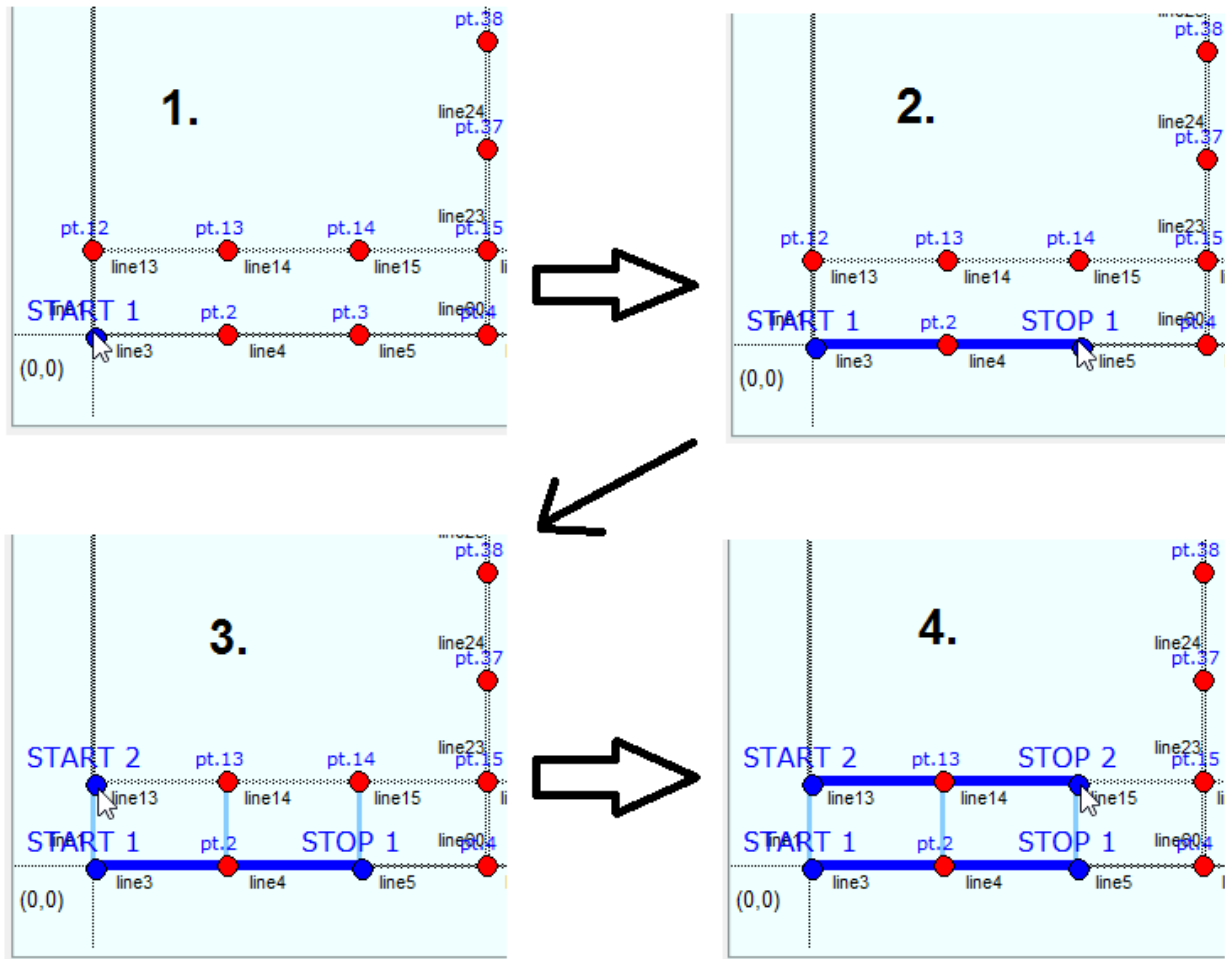
## Span optimization

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



The optimization is started as follows.

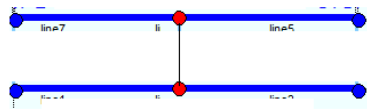
1. Start by clicking as illustrated next:



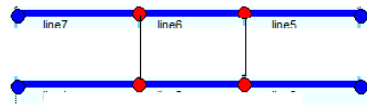
**2. Then run optimization.**

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.

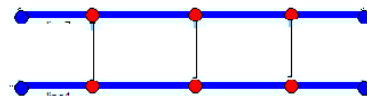
#### 4 plate fields



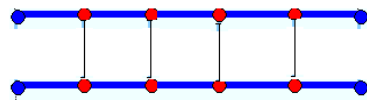
#### 6 plate fields



#### 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 thickenss	<input type="text" value="0"/>
Check for local stf. buckling	<input checked="" type="checkbox"/>	For weight calculation of girder: Max span mult / Min span mult	
		<input type="text" value="1.2"/>	<input type="text" value="0.8"/>
		Maximum span / Minimum span ->	<input type="text" value="6"/> <input type="text" value="2"/>

Results are presented as seen next.

RUN OPTIMIZATION!

anysmart

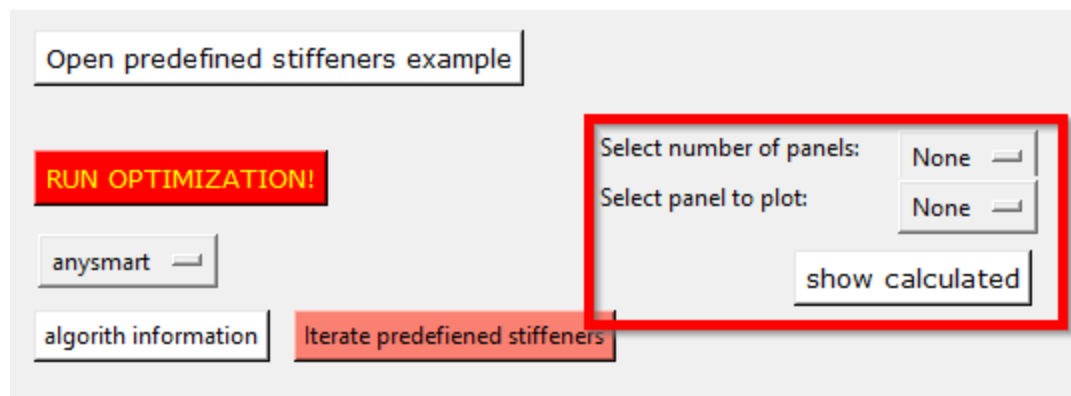
algorithm information

Results seen next. Weight index is tot\_weight / max\_weight  
max\_weight is the highest total weight of the checked variations.  
Weight index of 1 is the heaviest calculated variation.

Plate fields	Fields length	Weight index	All OK?
*****			
<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.

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

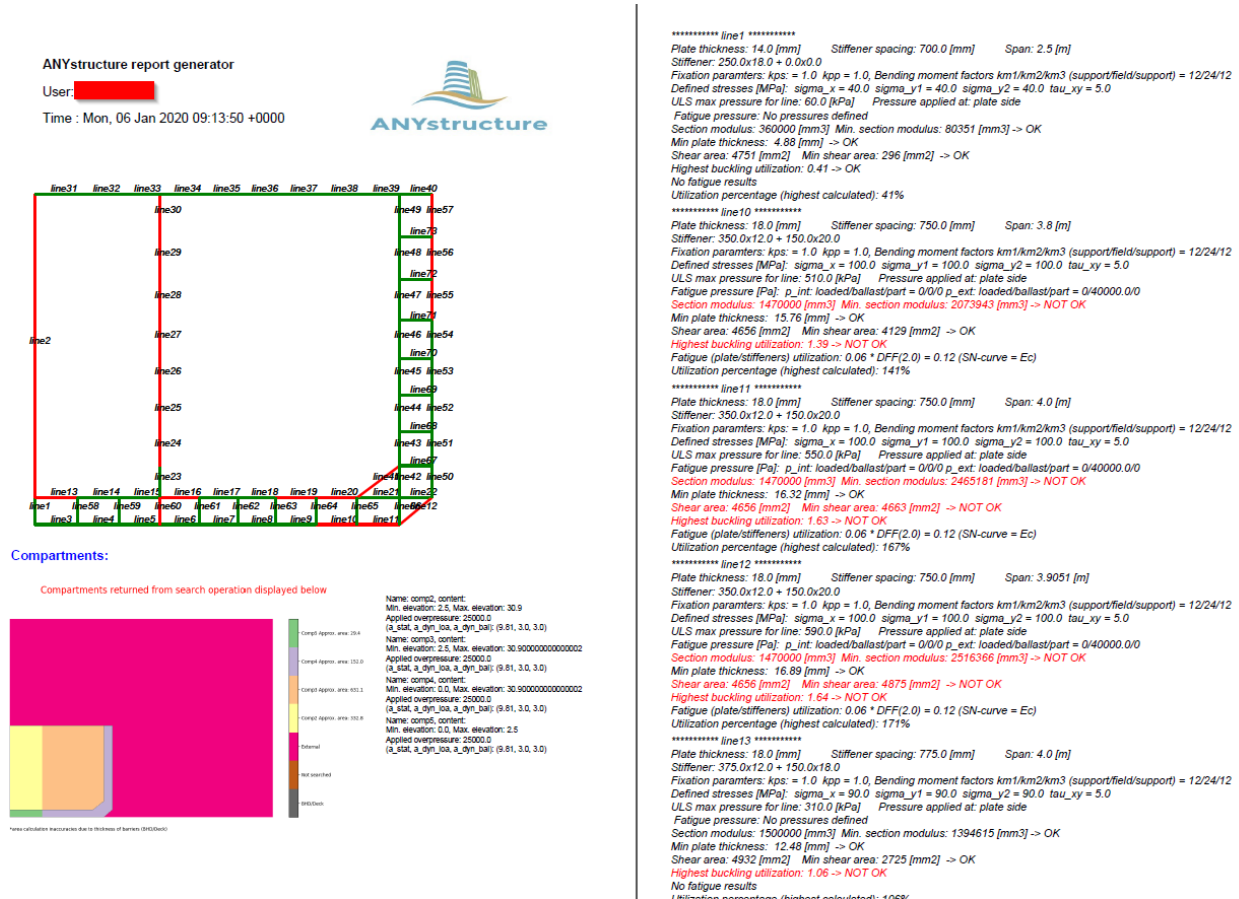


**Now close the window. Results are not currently returned to main window.**

**Detailed results, printed after running, looks like this :**

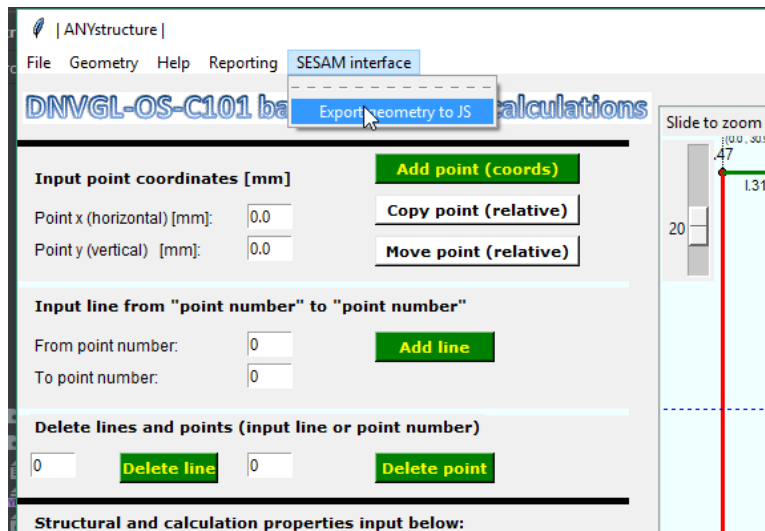
[illegible]

**A pdf report can be created by clicking “Reporting - Generate PDF report”. The report will include all information for all lines. An example is seen next.**



# 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:

