



# ANYstructure

## Documentation

The screenshot displays the ANYstructure software interface. The main window shows a 2D structural model of a tank with a grid of points and lines. The left-hand panel contains several input sections:

- Input point coordinates [mm]:** Includes fields for Point x (horizontal) [mm] and Point y (vertical) [mm], with 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 and To point number, with an "Add line" button.
- Delete lines and points:** Includes fields for Line number and Point number, with buttons for "Delete line", "Delete prop.", and "Delete point".
- Structural and calculation properties input below:** A table for defining material and structural properties.
- FLS:** Material yield [MPa] and Pressure side (p-plate, s-att.) options.
- Select structure type:** Includes a dropdown for "GENERAL\_INTERNAL\_WT" and an "Add structure to line" button.
- Find compartments and External pressures:** Buttons for finding compartments and external pressures.
- Comp. no.:** A list of compartments and buttons for "Display current compartments", "Set compartment properties", and "Delete all tanks".

The right-hand panel contains:

- Static and dynamic accelerations:** Input fields for Static acceleration [m/s<sup>2</sup>], Dyn. acc. loaded [m/s<sup>2</sup>], and Dyn. acc. ballast [m/s<sup>2</sup>], with a "Set accelerations" button.
- Optimize selected line/structure:** Includes "OPTIMIZE" and "SPAN" buttons.
- Combination for line (select line):** A dropdown menu for selecting line combinations.
- Names:** A table for defining names for different load cases.
- Pressures for this line:** A note about accounting for specific standards (DNV a/b, loaded/ballast, tank test, manual).
- Load info:** A button to view load information.

## 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>
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	17
<b>Reporting</b>	<b>22</b>
<b>Export to JS</b>	<b>23</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 shows a software interface for defining structure properties. It includes several panels and input fields:

- Define structure properties here --**: A central panel with a grid background showing a vertical beam structure. A red box labeled "Define plate and stiffener properties." points to the top flange area.
- Existing sections**: A diagram showing three different cross-sections of a beam with dimensions labeled (b1, h, y, z, tw, tf, b, t). A red box labeled "Existing sections" points to this diagram.
- Structural and calculation properties input below:**: A panel with various input fields for span, plate thickness, web height, web thickness, flange width, flange thickness, etc. A red box labeled "Define buckling calculation properties." points to the "Material yield" field.
- FLS**: A panel for Fatigue Limit State analysis, including material yield and structure type selection. A red box labeled "Define fatigue properties." points to the "Material yield" field.
- Define structure properties**: A panel with a dropdown menu for "type" and a list of existing sections: "Plate: 700.0x18.0", "Web: 400.0x12.0", and "Flange: 250.0x14.0".
- Diagram**: A 3D perspective view of a beam with a plate and stiffener, labeled "GIRDER length (Lg)" and "10".
- Buttons**: "Add line", "Delete line", "Add structure to line", and "Save and return structure".

## Display properties

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

l.pt.1 (0.0, 0.0) l.pt.2 (4.0, 0.0) l.pt.3 (7.5, 0.0) l.pt.4 (11.7, 0.0) l.pt.5 (16.4, 0.0) l.pt.6 (19.0, 0.0) l.pt.7 (22.0, 0.0) l.pt.8 (26.3, 0.0) l.pt.9 (30.1, 0.0) l.pt.10 (34.1, 0.0)

(0,0) 1.3 1.4 1.5 1.6 1.7 1.8 1.9 1.10 1.11

**Line 7**

**SELECTED: line7**

Applied compartments: Compartment 4

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  
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  
Pressure side (p-plate/s-stf): p

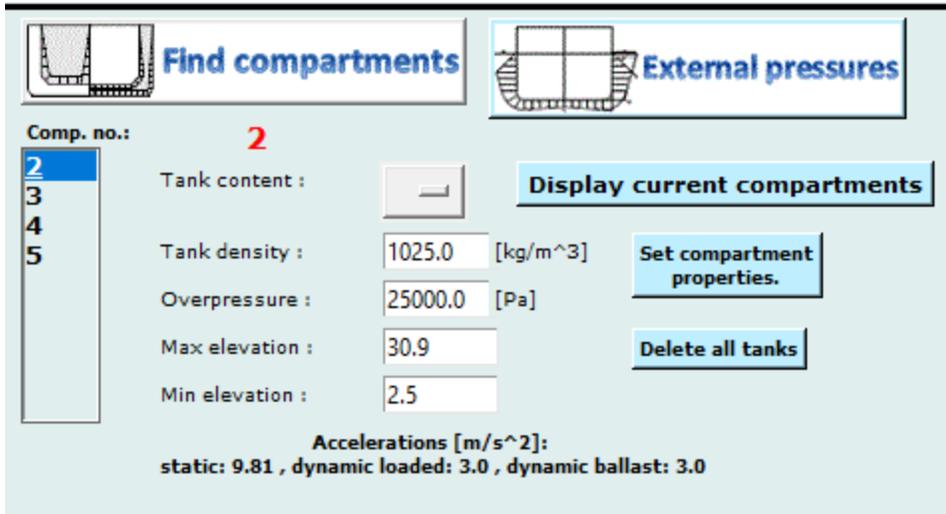
Applied static/dynamic loads: ballast\_bottom  
loaded\_static  
ballast\_static  
loaded\_bottom

## Define tanks

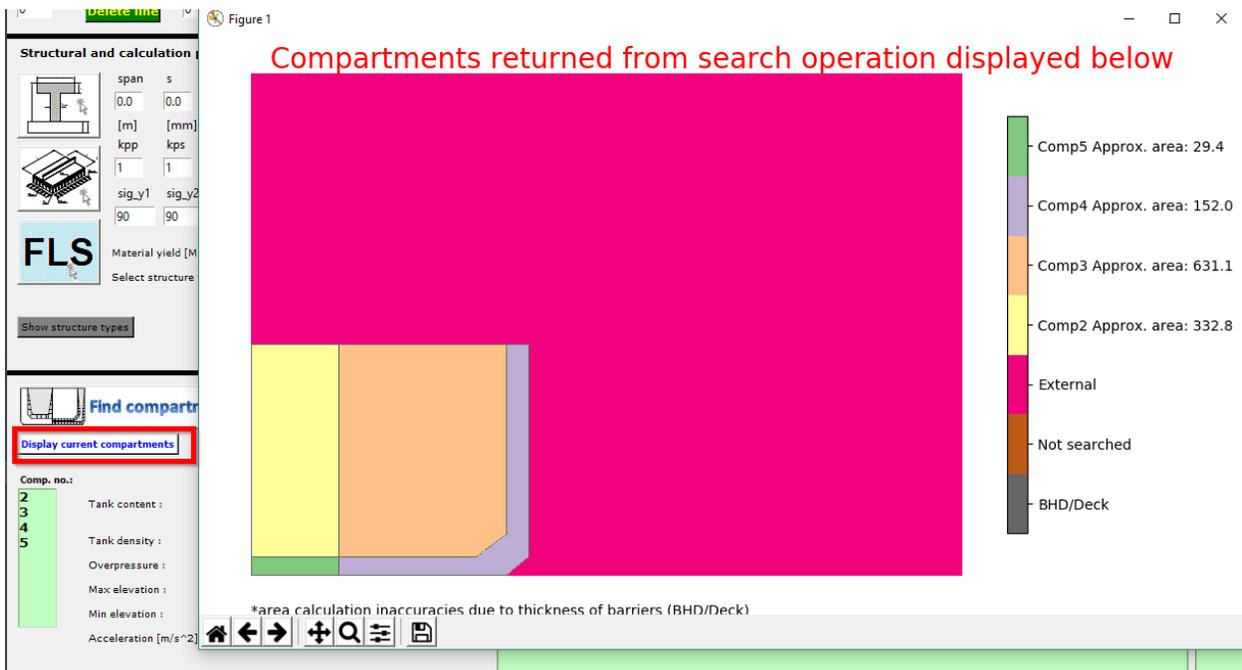
Tanks are searched for when clicking “Find compartments”. Non watertight structure are ignored. For information on structure types click “Show structure types”.

By default tank content density is set to 0.

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

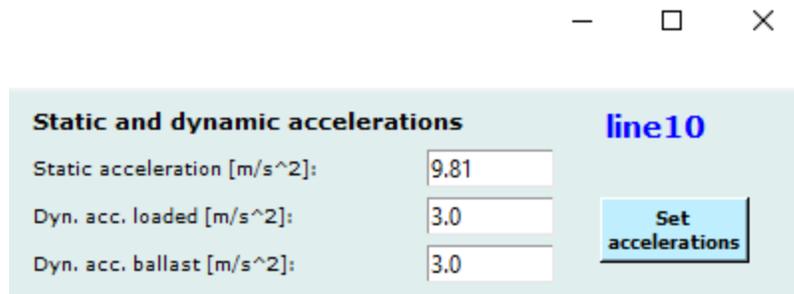


If you press “Display current compartments” after doing a compartment search, the result of the search is illustrated as seen next. Approximate area of the respective compartments is also shown.



## 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.

**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]

## 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

1

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"	1.3
Condition a) - Static load factor well defined loads	1.2
Condition a) - Dynamic load factor	0.7
Condition b) - Static load factor "unknown loads"	1
Condition b) - Static load factor well defined loads	1
Condition b) - Dynamic load factor	1.3
Tank test) - Static load factor "unknown loads"	1
Tank test) - Static load factor well defined loads	1
Tank test) - Dynamic load factor	0

**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.



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  
 $1 * 1.0 * 310707.2 + 25000.0 * 1 = 335707$   
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 not ok

Buckling not 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

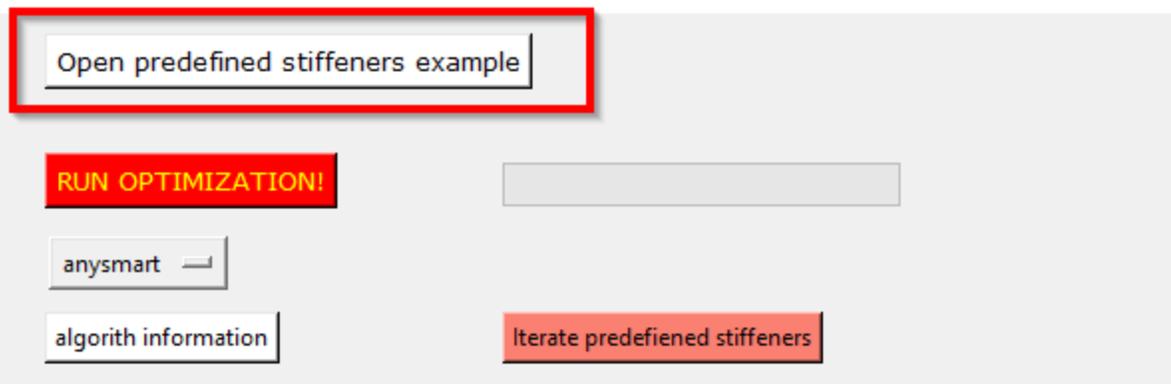
Section modulus: Wey1: 4.2400E+06 [mm<sup>3</sup>], Wey2: 1.4700E+06 [mm<sup>3</sup>]  
Minimum section modulus: 2.0739E+06 [mm<sup>3</sup>]  
Shear area: 4.6560E+03 [mm<sup>2</sup>]  
Minimum shear area: 4.1297E+03 [mm<sup>2</sup>]  
Plate thickness: 18.0 [mm]  
Minimum plate thickness: 15.8 [mm]  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

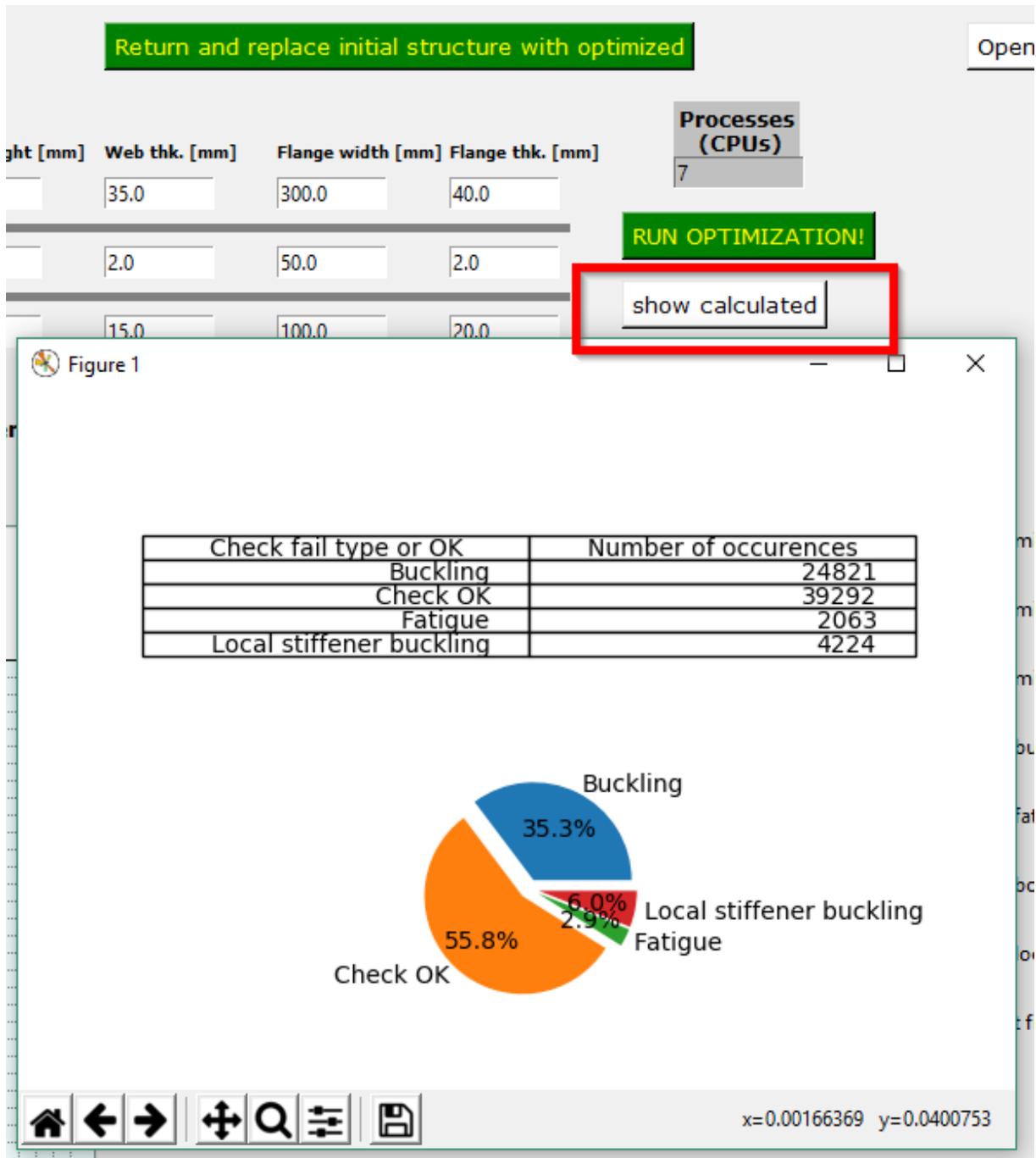
Various checks in the optimization module:

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

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

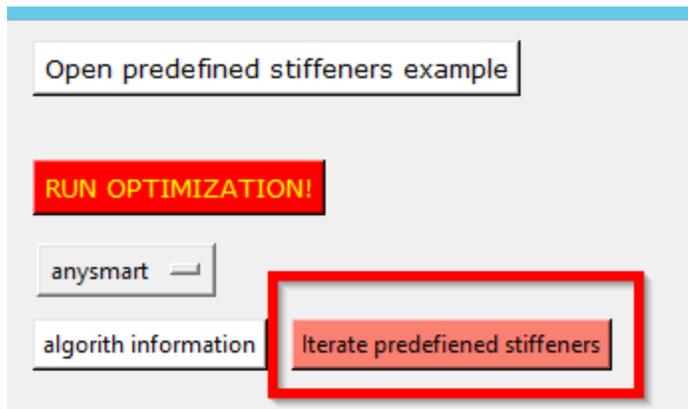
1. Same input on upper bounds, lower bounds and delta.
2. Click all the lines you want to include in the optimization.
3. Run the optimization.
4. Check the properties by **middle clicking** the line you ran.
5. If you are happy return the properties by clicking the top button

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

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

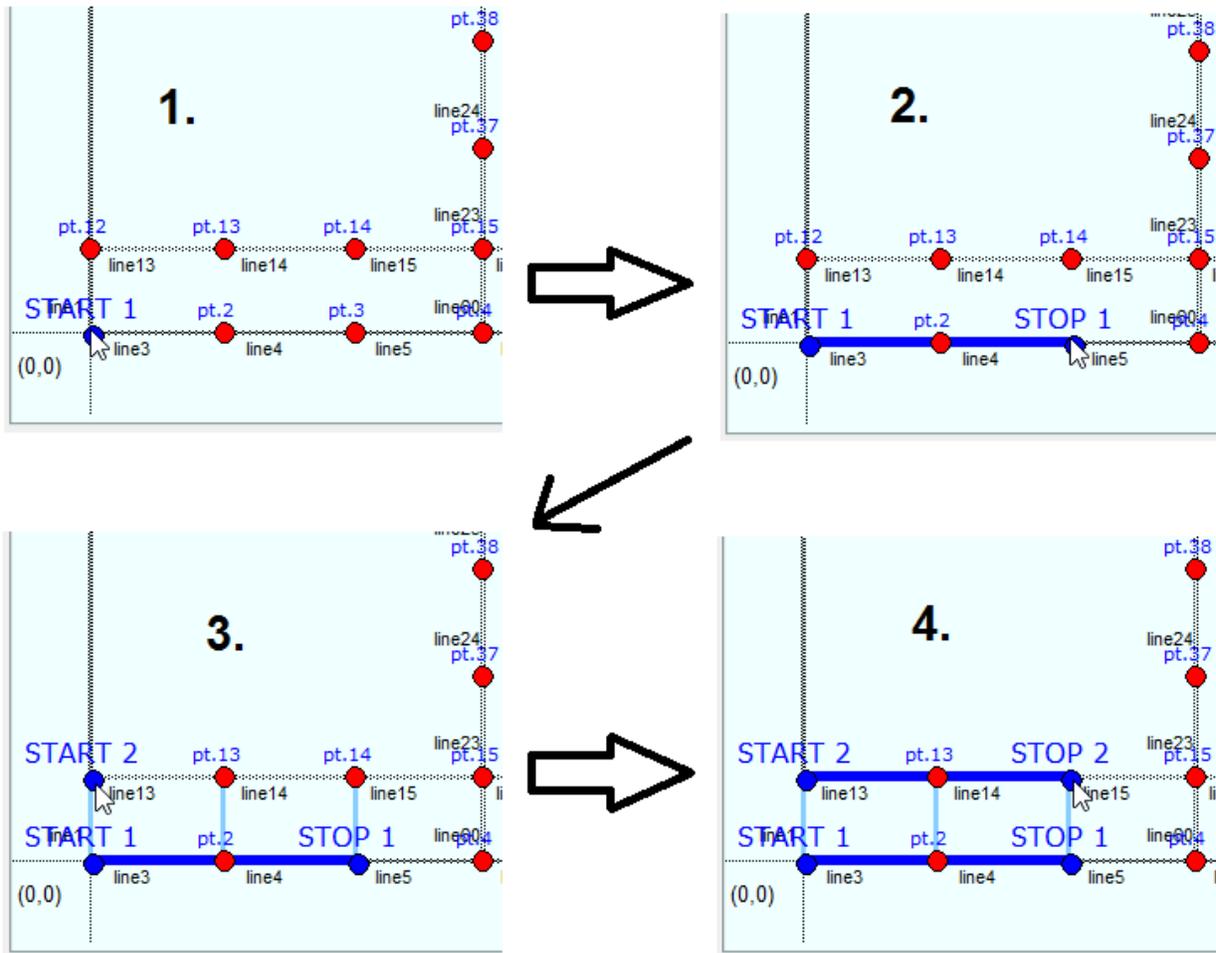
## Span optimization

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



The optimization is started as follows.

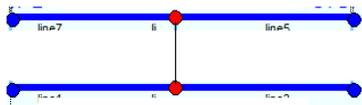
1. Start by clicking as illustrated next:



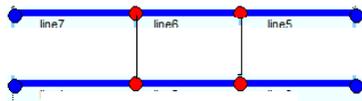
**2. Then run optimization.**

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.

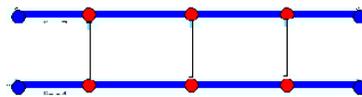
### 4 plate fields



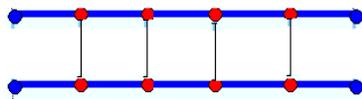
### 6 plate fields



### 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!

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

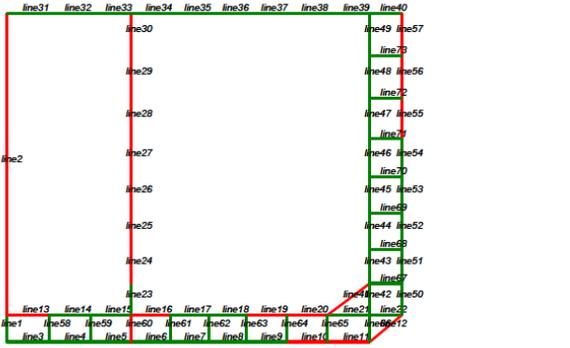
In this case 8 plate fields with length of 3 meter will give the lowest weight. 6 plate fields is almost equal.



# Reporting

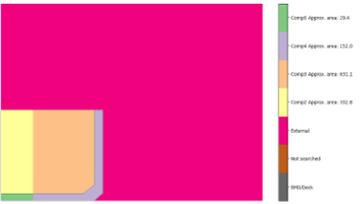
A pdf report can be created by clicking “Reporting - Generate PDF report”. The report will include all information for all lines. An example is seen next.

ANYstructure report generator  
 User: XXXXXXXXXX  
 Time : Mon, 06 Jan 2020 09:13:50 +0000

**Compartments:**

Compartments returned from search operation displayed below



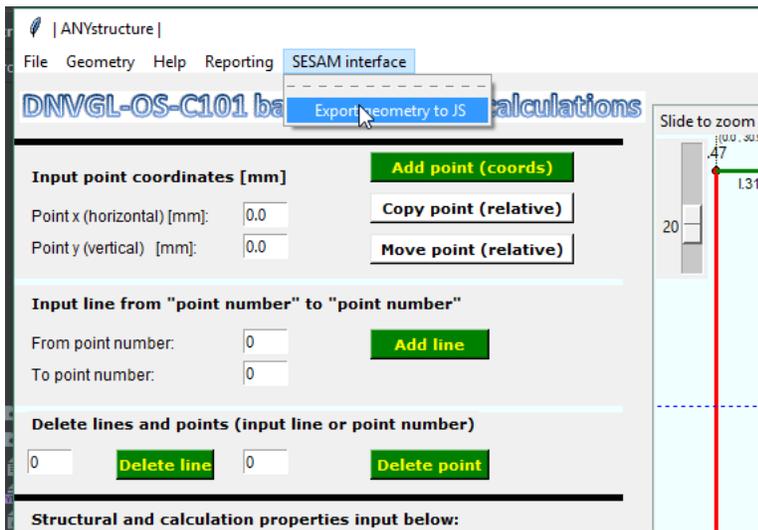
Compartment	Name	Min. elevation	Max. elevation
Comp1	comp1	2.5	30.9
Comp2	comp2	0.0	2.5
Comp3	comp3	0.0	2.5
Comp4	comp4	0.0	2.5

```

***** line1 *****
Plate thickness: 14.0 [mm]      Stiffener spacing: 700.0 [mm]      Span: 2.5 [m]
Stiffener: 250.0x18.0 + 0.0x0.0
Fixation parameters: kps = 1.0 kpp = 1.0, Bending moment factors km1/km2/km3 (support/field/support) = 12/24/12
Defined stresses [MPa]: sigma_x = 40.0 sigma_y1 = 40.0 sigma_y2 = 40.0 tau_xy = 5.0
ULS max pressure for line: 60.0 [kPa]      Pressure applied at: plate side
Fatigue pressure: No pressures defined
Section modulus: 360000 [mm3] Min. section modulus: 80351 [mm3] -> OK
Min plate thickness: 4.88 [mm] -> OK
Shear area: 4751 [mm2]      Min shear area: 296 [mm2] -> OK
Highest buckling utilization: 0.41 -> OK
No fatigue results
Utilization percentage (highest calculated): 41%
***** line10 *****
Plate thickness: 18.0 [mm]      Stiffener spacing: 750.0 [mm]      Span: 3.8 [m]
Stiffener: 350.0x12.0 + 150.0x20.0
Fixation parameters: kps = 1.0 kpp = 1.0, Bending moment factors km1/km2/km3 (support/field/support) = 12/24/12
Defined stresses [MPa]: sigma_x = 100.0 sigma_y1 = 100.0 sigma_y2 = 100.0 tau_xy = 5.0
ULS max pressure for line: 510.0 [kPa]      Pressure applied at: plate side
Fatigue pressure [Pa]: p_int: loaded/ballast/part = 0/0/0 p_ext: loaded/ballast/part = 0/40000.0/0
Section modulus: 1470000 [mm3] Min. section modulus: 2073943 [mm3] -> NOT OK
Min plate thickness: 15.76 [mm] -> OK
Shear area: 4656 [mm2]      Min shear area: 4129 [mm2] -> OK
Highest buckling utilization: 1.39 -> NOT OK
Fatigue (plate/stiffeners) utilization: 0.06 * DFF(2,0) = 0.12 (SN-curve = Ec)
Utilization percentage (highest calculated): 141%
***** line11 *****
Plate thickness: 18.0 [mm]      Stiffener spacing: 750.0 [mm]      Span: 4.0 [m]
Stiffener: 350.0x12.0 + 150.0x20.0
Fixation parameters: kps = 1.0 kpp = 1.0, Bending moment factors km1/km2/km3 (support/field/support) = 12/24/12
Defined stresses [MPa]: sigma_x = 100.0 sigma_y1 = 100.0 sigma_y2 = 100.0 tau_xy = 5.0
ULS max pressure for line: 550.0 [kPa]      Pressure applied at: plate side
Fatigue pressure [Pa]: p_int: loaded/ballast/part = 0/0/0 p_ext: loaded/ballast/part = 0/40000.0/0
Section modulus: 1470000 [mm3] Min. section modulus: 2465181 [mm3] -> NOT OK
Min plate thickness: 16.32 [mm] -> OK
Shear area: 4656 [mm2]      Min shear area: 4663 [mm2] -> NOT OK
Highest buckling utilization: 1.63 -> NOT OK
Fatigue (plate/stiffeners) utilization: 0.06 * DFF(2,0) = 0.12 (SN-curve = Ec)
Utilization percentage (highest calculated): 167%
***** line12 *****
Plate thickness: 18.0 [mm]      Stiffener spacing: 750.0 [mm]      Span: 3.9051 [m]
Stiffener: 350.0x12.0 + 150.0x20.0
Fixation parameters: kps = 1.0 kpp = 1.0, Bending moment factors km1/km2/km3 (support/field/support) = 12/24/12
Defined stresses [MPa]: sigma_x = 100.0 sigma_y1 = 100.0 sigma_y2 = 100.0 tau_xy = 5.0
ULS max pressure for line: 590.0 [kPa]      Pressure applied at: plate side
Fatigue pressure [Pa]: p_int: loaded/ballast/part = 0/0/0 p_ext: loaded/ballast/part = 0/40000.0/0
Section modulus: 1470000 [mm3] Min. section modulus: 2516366 [mm3] -> NOT OK
Min plate thickness: 16.89 [mm] -> OK
Shear area: 4656 [mm2]      Min shear area: 4875 [mm2] -> NOT OK
Highest buckling utilization: 1.64 -> NOT OK
Fatigue (plate/stiffeners) utilization: 0.06 * DFF(2,0) = 0.12 (SN-curve = Ec)
Utilization percentage (highest calculated): 171%
***** line13 *****
Plate thickness: 18.0 [mm]      Stiffener spacing: 775.0 [mm]      Span: 4.0 [m]
Stiffener: 375.0x12.0 + 150.0x18.0
Fixation parameters: kps = 1.0 kpp = 1.0, Bending moment factors km1/km2/km3 (support/field/support) = 12/24/12
Defined stresses [MPa]: sigma_x = 90.0 sigma_y1 = 90.0 sigma_y2 = 90.0 tau_xy = 5.0
ULS max pressure for line: 310.0 [kPa]      Pressure applied at: plate side
Fatigue pressure: No pressures defined
Section modulus: 1500000 [mm3] Min. section modulus: 1394615 [mm3] -> OK
Min plate thickness: 12.48 [mm] -> OK
Shear area: 4932 [mm2]      Min shear area: 2725 [mm2] -> OK
Highest buckling utilization: 1.06 -> NOT OK
No fatigue results
Utilization percentage (highest calculated): 106%
        
```

# 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:

