



ANYstructure

Documentation

ANYstructure | ship_section_example.txt

File Geometry Reporting SESAM interface Help

Input point coordinates [mm]

Point x (horizontal) [mm]: 0.0 **Add point (coords)**

Point y (vertical) [mm]: 0.0 **Copy 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_th web_thk fl_w fl_thk

(m) (mm) (mm) (mm) (mm) (mm)

kpp kps km1 km2 k3

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.)

Add structure to line

Find compartments

Comp. no.: 2

Tank content: [] **Display current compartments**

Tank density: 0 [kg/m³] **Set compartment properties.**

Overpressure: 25000 [Pa] **Delete all tanks**

Max elevation: 0.0

Min elevation: 0.0

Acceleration [m/s²]:

External pressures

Static and dynamic accelerations

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

1

Name: Stat. LF Dyn. LF Include?

ballast_bottom 0.0 0.7 ☒

loaded_static 1.3 0.0 ☒

ballast_static 1.3 0.0 ☒

loaded_bottom 0.0 0.7 ☒

Compartment4 1.2 0.7 ☒

Manual (pressure/LF) 0.0 1.0 ☒

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**

House left click: select line
House right click: select point

loaded_static [m]

ballast_static [m]

pt.47 pt.48 pt.49 pt.44 pt.50 pt.51 pt.52 pt.53 pt.54 pt.45 pt.46

131 132 133 134 135 136 137 138 139 140

129 128 127 126 125 124 123 122 121 120

117 116 115 114 113 112 111 110 109 108 107 106 105 104 103 102 101 100

99 98 97 96 95 94 93 92 91 90 89 88 87 86 85 84 83 82 81 80 79 78 77 76 75 74 73 72 71 70 69 68 67 66 65 64 63 62 61 60 59 58 57 56 55 54 53 52 51 50 49 48 47

13 14 15 16 17 18 19 20 21 22 23 24 25 26 27 28 29 30 31 32 33 34 35 36 37 38 39 40 41 42 43 44 45 46 47 48 49 50 51 52 53 54 55 56 57 58 59 60 61 62 63 64 65 66 67 68 69 70 71 72 73 74 75 76 77 78 79 80 81 82 83 84 85 86 87 88 89 90 91 92 93 94 95 96 97 98 99 100

2021 Version 2.X

Table of contents

Table of contents	2
Theory	3
Modelling	4
Assigning properties	5
Display properties	6
Define tanks	6
Setting accelerations	8
Define external pressures	8
Load combinations	9
Changing load factors	10
Reviewing loads	11
Results	13
Optimization	13
Optimization iteration by predefined stiffeners	13
Single optimization	14
Multiple optimization	17
Span optimization	18
Reporting	23
Export to JS	24

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 DNV-GL logo is displayed in a bold, dark blue, sans-serif font. The letters are widely spaced, and the logo is positioned to the right of the decorative horizontal bars.

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)

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

Speed up your modelling significantly by using the shortcuts:

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

- 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:

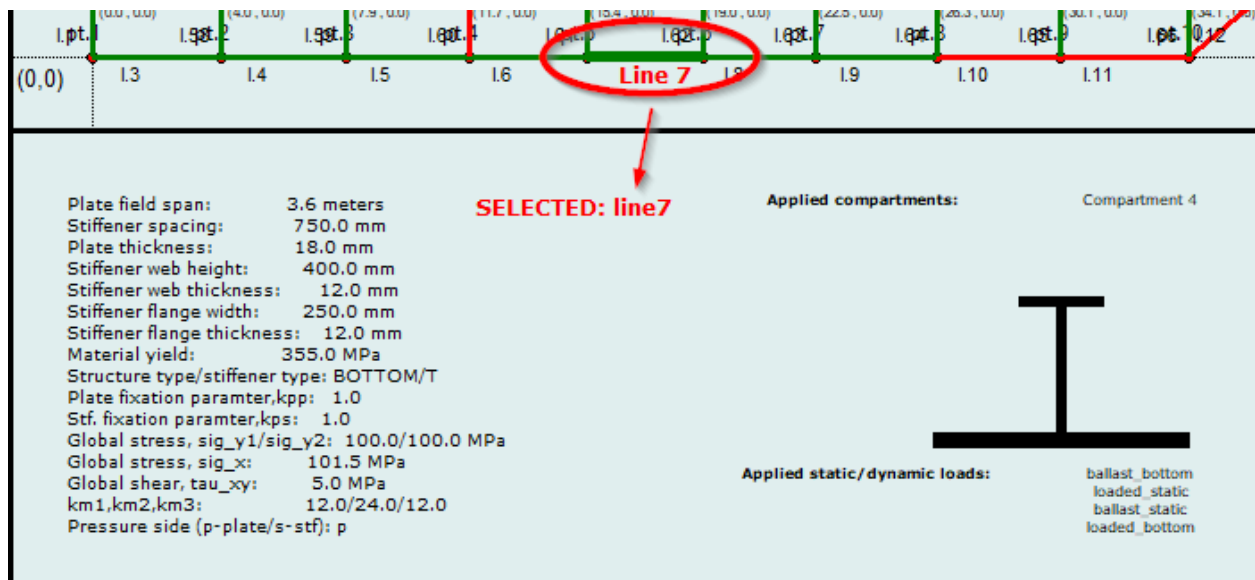
- Define plate and stiffener properties.**: Points to the 'Add line' button and the 'Delete line' button in the top left section.
- Existing sections**: Points to the 'Existing sections' dropdown menu in the middle right section.
- Define buckling calculation properties.**: Points to the 'Find compartments' button in the bottom left section.
- Define fatigue properties.**: Points to the 'Material yield [MPa]' field in the bottom left section.

The interface includes the following sections:

- Input line from "point number" to "point number"**: Fields for 'From point number' and 'To point number'.
- Delete lines and points**: A 'Delete line' button.
- Structural and calculation properties input below:** A table for inputting properties like span, plate thickness, web height, etc.
- Define structure properties here --**: A section for defining the structure type and dimensions (Plate thk., Web height, Web thk., Flange width, Flange thk.).
- Existing sections:** A dropdown menu to select from previously defined sections.
- Plate: 700.0x18.0, Web: 400.0x12.0, Flange: 250.0x14.0**: A summary of the current structure properties.
- Find compartments**: A section for defining compartments and their properties.
- Save and return structure**: A green button at the bottom right.

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.

Find compartments

Comp. no.: **2**

2
3
4
5

Tank content :

Tank density : [kg/m³]

Overpressure : [Pa]

Max elevation :

Min elevation :

Display current compartments

Set compartment properties.

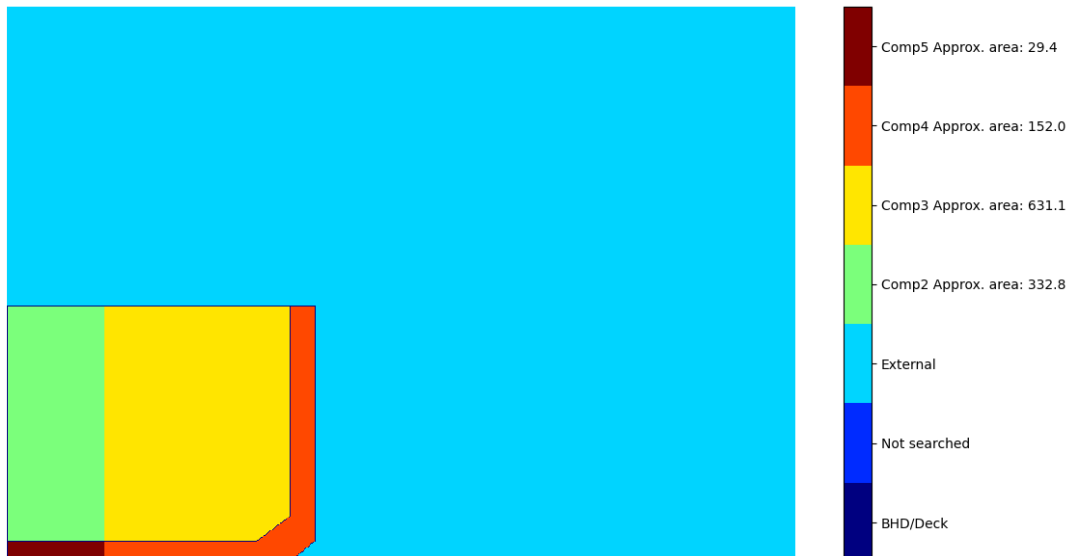
Delete all tanks

Accelerations [m/s²]:
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.



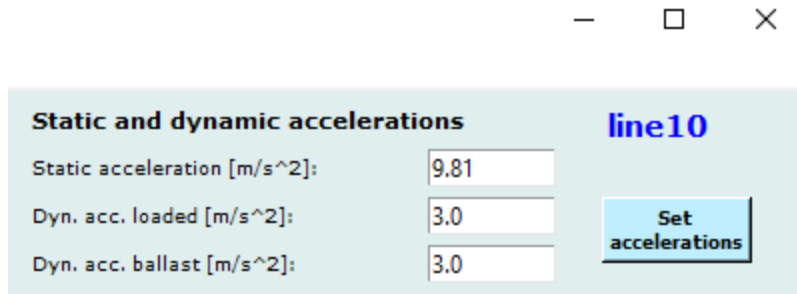
Compartments returned from search operation displayed below



*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.



Static and dynamic accelerations line10

Static acceleration [m/s ²]:	9.81
Dyn. acc. loaded [m/s ²]:	3.0
Dyn. acc. ballast [m/s ²]:	3.0

Set accelerations

Define external pressures

Click “External pressures” to define pressures acting on the structures.

NOTE:

FOR DYNAMIC EQUATION THE FOLLOWING APPLIES

X (horizontal) used for BOTTOM, BBT, HOPPER, MD

Y (vertical) used for BBS, SIDE_SHELL, SSS

After new window is opened:

- 1. Make dynamic loads**
 - a. Dynamic loads are made by defining up to 3rd degree equations. X or Y direction depends on the defined structure type.**
 - b. Note that you can define a constant dynamic load by using Constant (Constant (C)) only.**
- 2. Static loads are calculated according to depth.**

3. To apply a defined load to a line or multiple lines:
 - a. a. Select load by clicking the created load
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.

Load properties

1. Dynamic loads

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

Input load name:

Third degree poly [x^3]:

Second degree poly [x^2]:

First degree poly [x]:

Constant [C]:

Load condition:

Limit state:

Create dynamic load

2. Static loads

Hydrostatic loads defined by draft.

Define name of static load:

Define static draft from sea:

Select load condition:

Create static load

3. Slamming pressure

Load name:

Pressure [Pa]:

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 ----->

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

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.

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

1

Line:	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 γ_1 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 $\gamma_1 = 1.0$ 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 γ_1 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:
 $1 \cdot 1.3 \cdot 221215.5 = 287580.2$
 dynamic with acceleration: 3.0 is:
 $1 \cdot 0.7 \cdot 198687.0 = 139080.9$

 RESULT: $287580.2 + 139081 = 426661.1$

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

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

 RESULT: $43561.7 + 196077 = 239639.0$

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

 RESULT: $221215.5 + 258293 = 479508.7$

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

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

 RESULT: $80900.2 + 150829 = 231729.0$

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

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^3], Wey2: 1.7500E+06 [mm^3]
Minimum section modulus: 1.7163E+06 [mm^3]

Shear area: 5.1600E+03 [mm^2]
Minimum shear area: 3.5296E+03 [mm^2]

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^3], Wey2: 1.4700E+06 [mm^3]
Minimum section modulus: 2.0739E+06 [mm^3]

Shear area: 4.6560E+03 [mm^2]
Minimum shear area: 4.1297E+03 [mm^2]

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

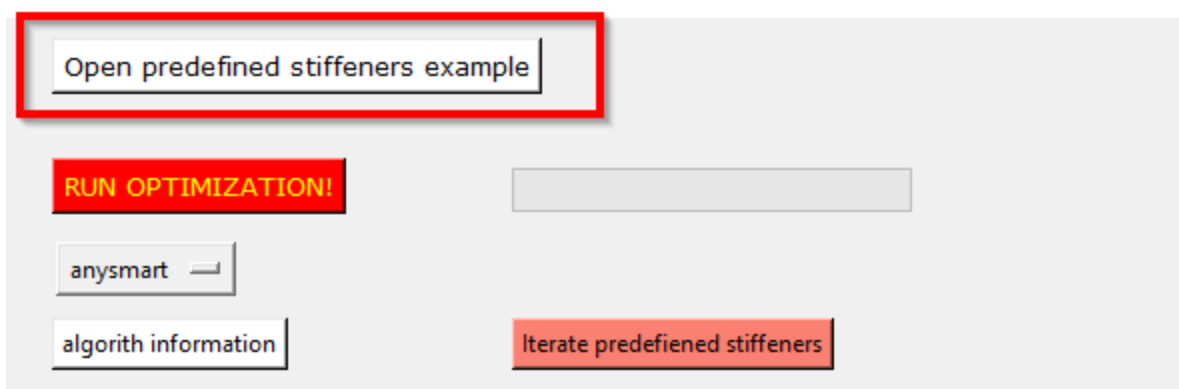
Fatigue results (DNVGL-RP-C203):
Total damage (DFF not included): 0.058 | With DFF = 2.0 --> Damage: 0.117
```

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

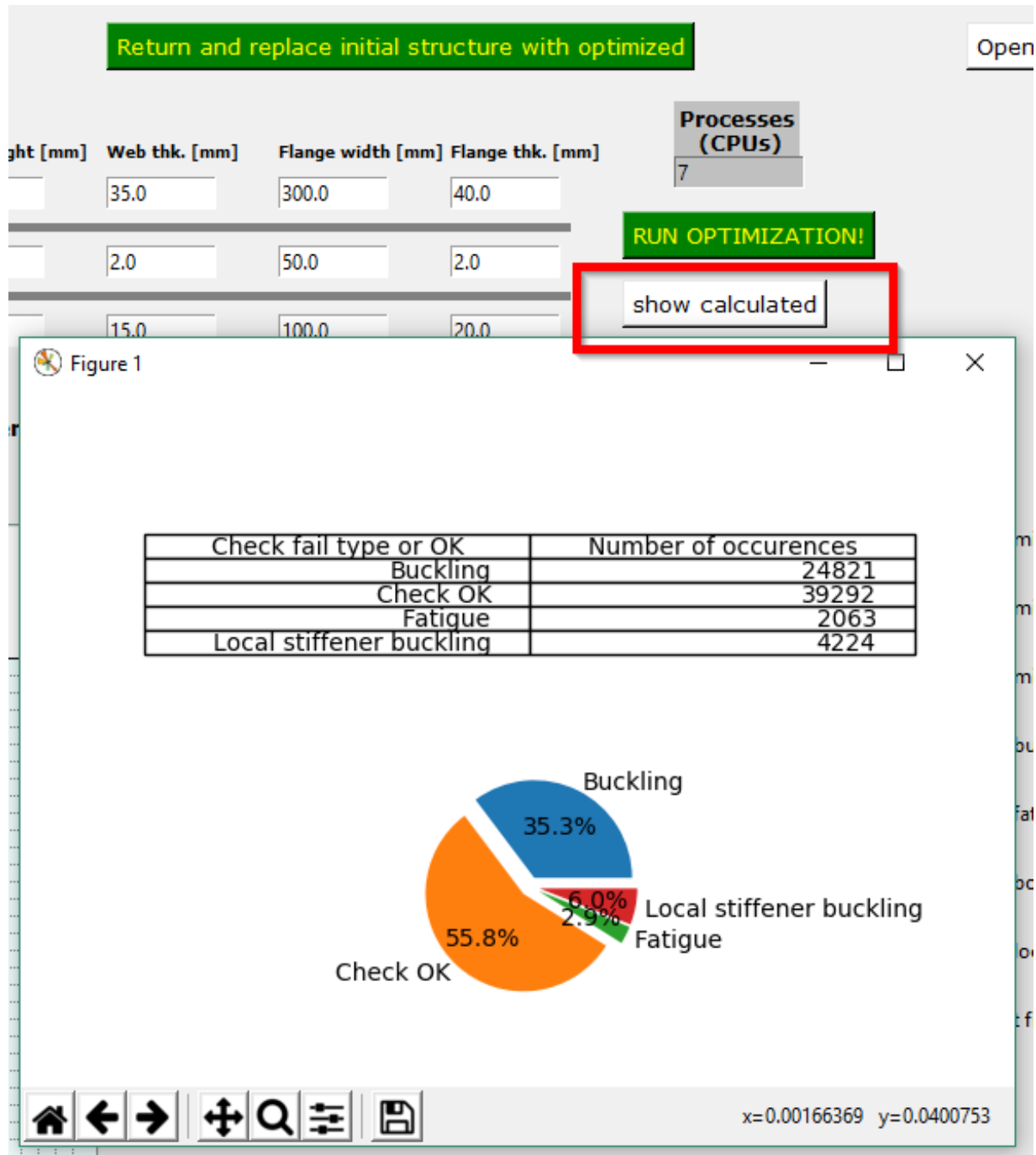
Various checks in the optimization module:

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.

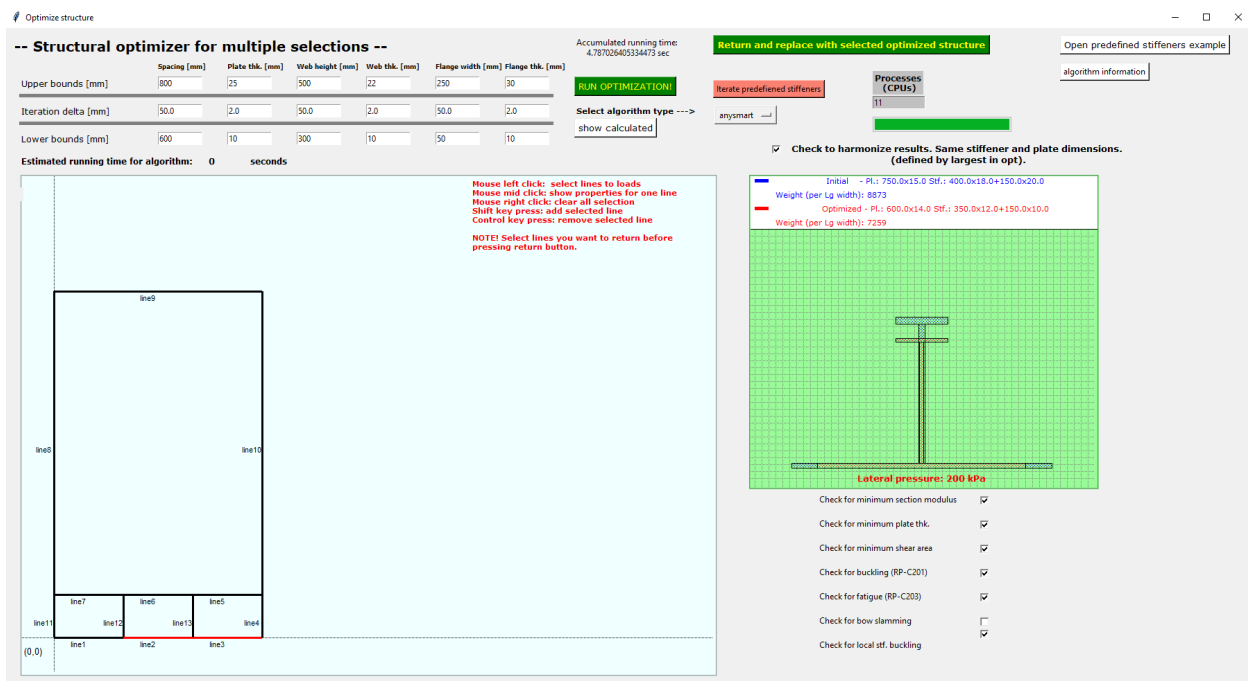
Check for minimum section modulus	<input checked="" type="checkbox"/>
Check for minimum plate thk.	<input checked="" type="checkbox"/>
Check for minimum shear area	<input checked="" type="checkbox"/>
Check for buckling (RP-C201)	<input checked="" type="checkbox"/>
Check for fatigue (RP-C203)	<input checked="" 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



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.

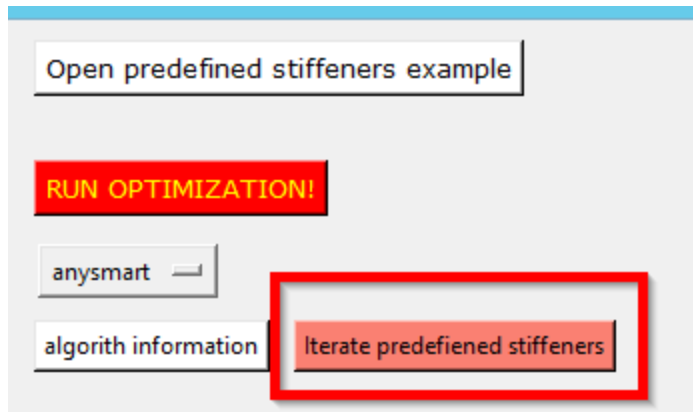
The optimization can be **harmonized**. That means that the largest dimension found in the multiple optimization is used for all selected. This is done after all plates/stiffeners are checked. Harmonization can only be done in the multiopt option

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

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

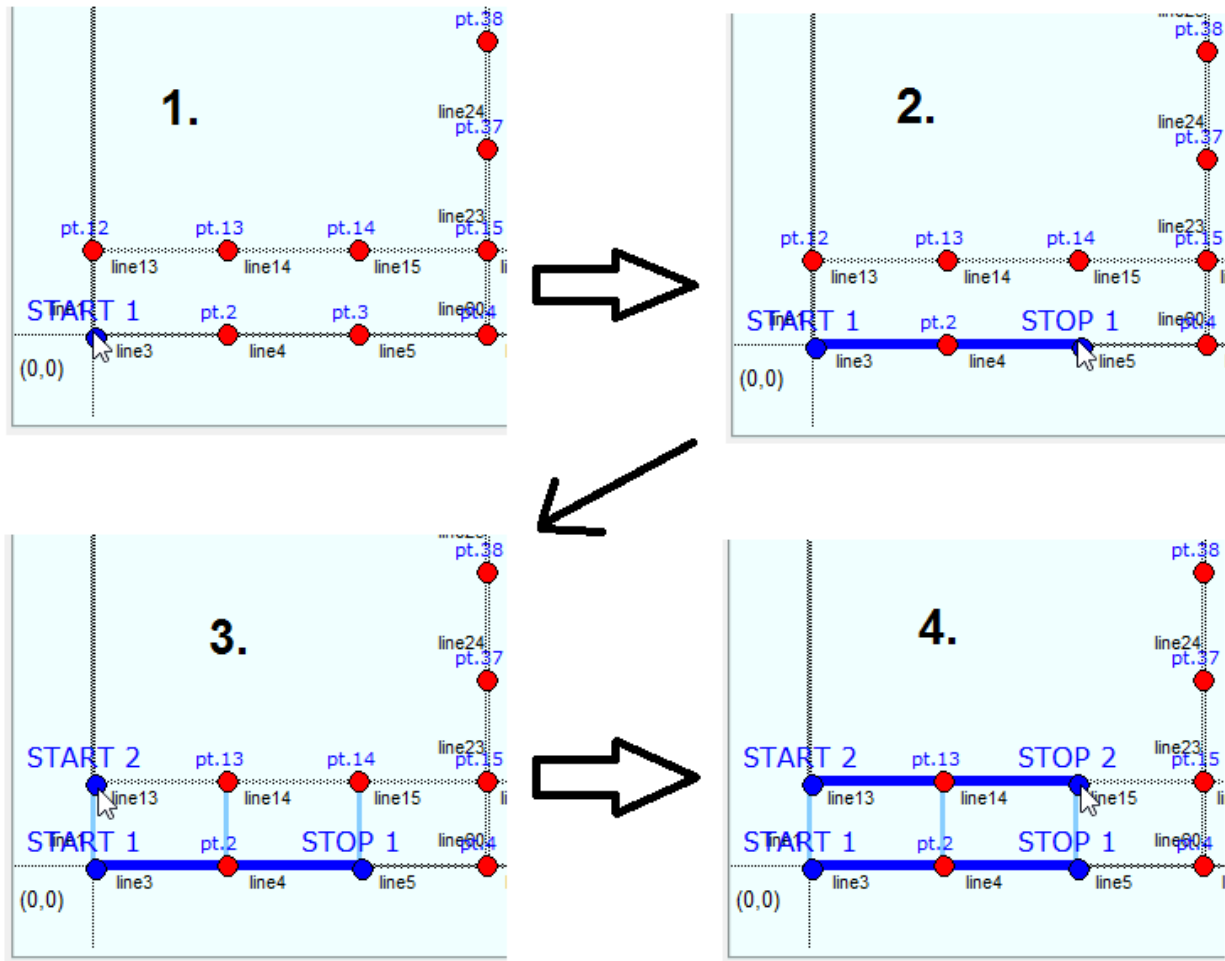
Span optimization

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



The optimization is started as follows.

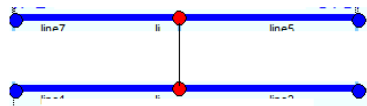
1. Start by clicking as illustrated next:



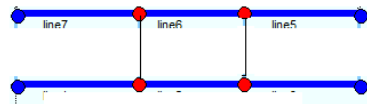
2. Then run optimization.

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.

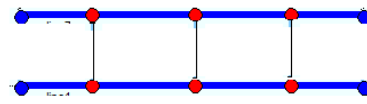
4 plate fields



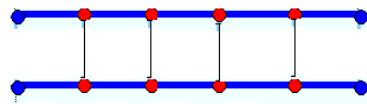
6 plate fields



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"/>	Frame (girder data) for weight calculation:	
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.

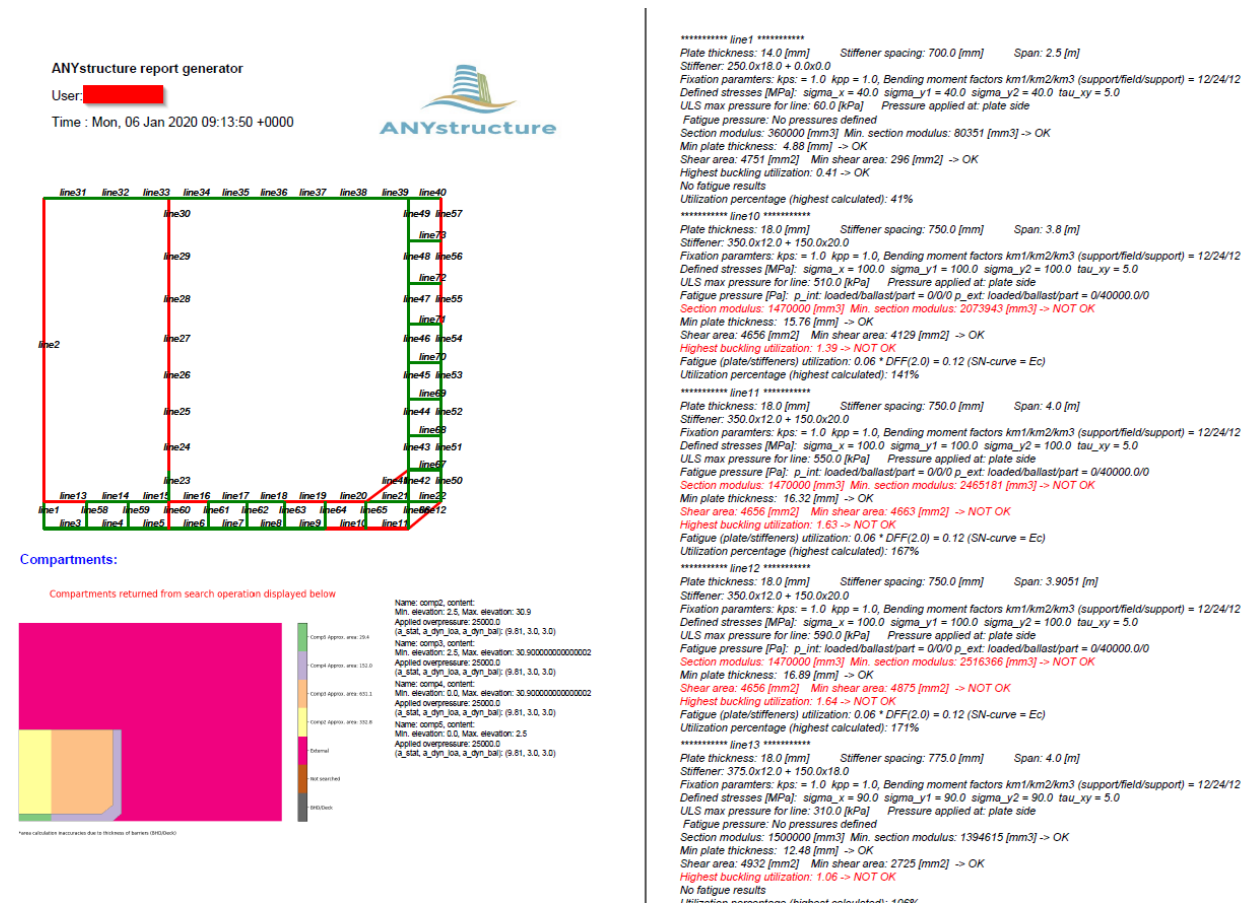
Plate fields	Fields length	Weight index	All OK?

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.

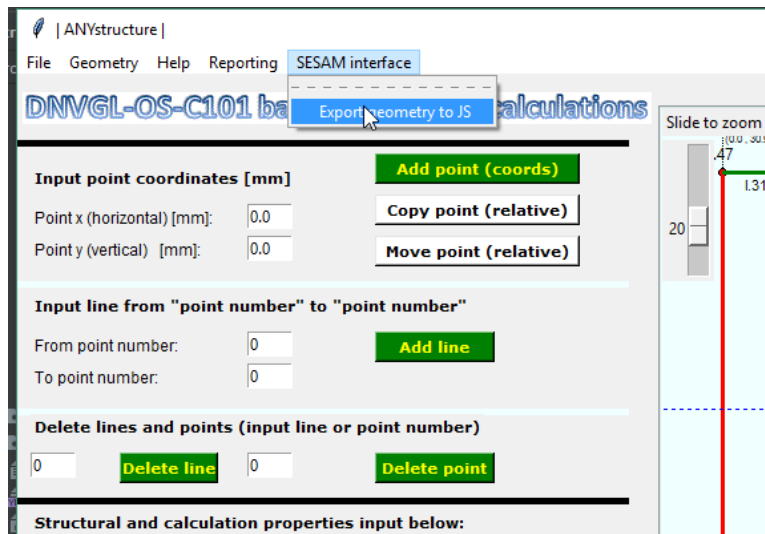
Reporting

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.



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:

