



ANYstructure

File Geometry Reporting SESAM interface Help

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

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

Delete lines and points (or left/right click and use "Delete key")
 Line number (left click): 7 **Delete line** **Delete prop.**
 Point number (right click): 0 **Delete point**

Structural and calculation properties input below:
 span s pl.thick 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**

Comp. no.:
 2 Tank content: **Display current compartments**
 3 Tank density: 0 [kg/m³] **Set compartment properties.**
 4 Overpressure: 25000 [Pa] **Delete all tanks**
 5 Max elevation: 0.0
 Min elevation: 0.0
 Acceleration [m/s²]:

Check to see available shortcuts

CTRL-Z Undo geometry action
 CTRL-C Copy selected point
 CTRL-M Move selected point
 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
 Arrows left/right - previous/next line

Static and dynamic accelerations **line7**
 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)
 DNV a [Pa]: {403219, 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: > horizontal
 Plate fixation parameter:kpp: 1.0
 Global stress_s1g_y1/s1g_y2: 100.0/100.0 MPa
 Global stress_s1g_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

SELECTED: line7 Applied compartments: Compartment 4

Applied static/dynamic loads: ballast_bottom, loaded_static, ballast_static, loaded_bottom

Section modulus: Wey1: 4.8300E+06 [mm³], Wey2: 1.7500E+06 [mm³]
 Minimum section modulus: 1.7162E+06 [mm³]

Shear area: 5.1600E+03 [mm²]
 Minimum shear area: 3.5296E+03 [mm²]

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

Buckling results DNV-RP-C201:
 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

Documentation

2021 Version 2.X

Table of contents

Table of contents	2
Theory	3
Modelling	4
Assigning properties	5
Display properties	6
Define tanks	7
Setting accelerations	8
Define external pressures	9
Load combinations	10
Changing load factors	11
Reviewing loads	12
Results	14
Optimization	14
Optimization iteration by predefined stiffeners	14
Single optimization	15
Multiple optimization	18
Span optimization	19
Reporting	24
Export to JS	25

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)

Input point coordinates [mm]	<input type="text"/>	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-P	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

Left and right arrow to change current line.

Assigning properties

Input properties manually or click the button indicated below to set the values.

Values are set by clicking “Add structure to line”. This also applies to fatigue properties. If you have added a property to a line and want to use the same for the next line, just press “Add structure to line” on the new line.

All beam sections are recorded. If you want to apply an existing, choose it from the drop down menu. Then press “Save and return structure”.

The screenshot displays a software interface for defining structure properties. The main window is titled "Define structure properties" and contains several sections:

- Input line from "point number" to "point number"**: Fields for "From point number" (0) and "To point number" (4), with "Add line" and "Delete line" buttons.
- Structural and calculation properties input below:** A table with columns for span, plate thickness, web height, web thickness, flange width, flange thickness, and various stiffness and strength parameters. A "FLS" button is visible.
- Define plate and stiffener properties:** A section with input fields for Plate thk., Web height, Web thk., Flange width, and Flange thk., along with a diagram of a beam section and a "Save and return structure" button.
- Existing sections:** A dropdown menu showing "Plate: 700.0x18.0", "Web: 400.0x12.0", and "Flange: 250.0x14.0".
- Define buckling calculation properties:** A section with input fields for Tank content, Tank density, and Overpressure, with a "Delete all tank" button.
- Define fatigue properties:** A section with input fields for Material yield and a "Show structure types" button.

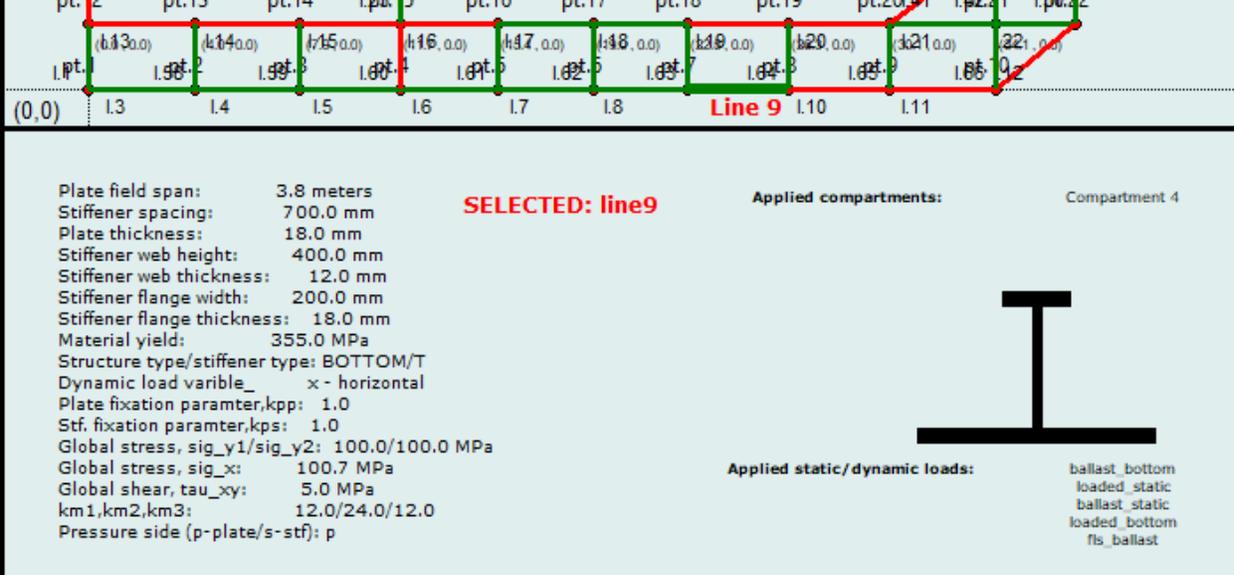
Red callout boxes highlight specific areas: "Define plate and stiffener properties." points to the input fields; "Define buckling calculation properties." points to the Tank content field; and "Define fatigue properties." points to the Material yield field. A diagram of a beam section is labeled "Existing sections" and "Girder length (Lg)".

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.



The screenshot shows a top-down view of a plate with stiffeners. The plate is divided into compartments, with 'Line 9' highlighted in red. The properties window for 'Line 9' is displayed below the plate view.

Property	Value	Applied compartments	Value
Plate field span:	3.8 meters	Applied compartments:	Compartment 4
Stiffener spacing:	700.0 mm		
Plate thickness:	18.0 mm		
Stiffener web height:	400.0 mm		
Stiffener web thickness:	12.0 mm		
Stiffener flange width:	200.0 mm		
Stiffener flange thickness:	18.0 mm		
Material yield:	355.0 MPa		
Structure type/stiffener type:	BOTTOM/T		
Dynamic load variable_	x - horizontal		
Plate fixation parameter,kpp:	1.0		
Stf. fixation parameter,kps:	1.0		
Global stress, sig_y1/sig_y2:	100.0/100.0 MPa		
Global stress, sig_x:	100.7 MPa	Applied static/dynamic loads:	ballast_bottom loaded_static ballast_static loaded_bottom fls_ballast
Global shear, tau_xy:	5.0 MPa		
km1,km2,km3:	12.0/24.0/12.0		
Pressure side (p-plate/s-stf):	p		

Define tanks

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

By default tank content density is set to 0.

After 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 : fresh water

Tank density : 1000 [kg/m³]

Overpressure : 25000.0 [Pa]

Max elevation : 30.9

Min elevation : 2.5

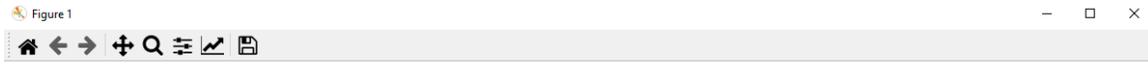
Accelerations [m/s²]:
static: 9.81 , dynamic loaded: 3.0 , dynamic ballast: 3.0

Display current compartments

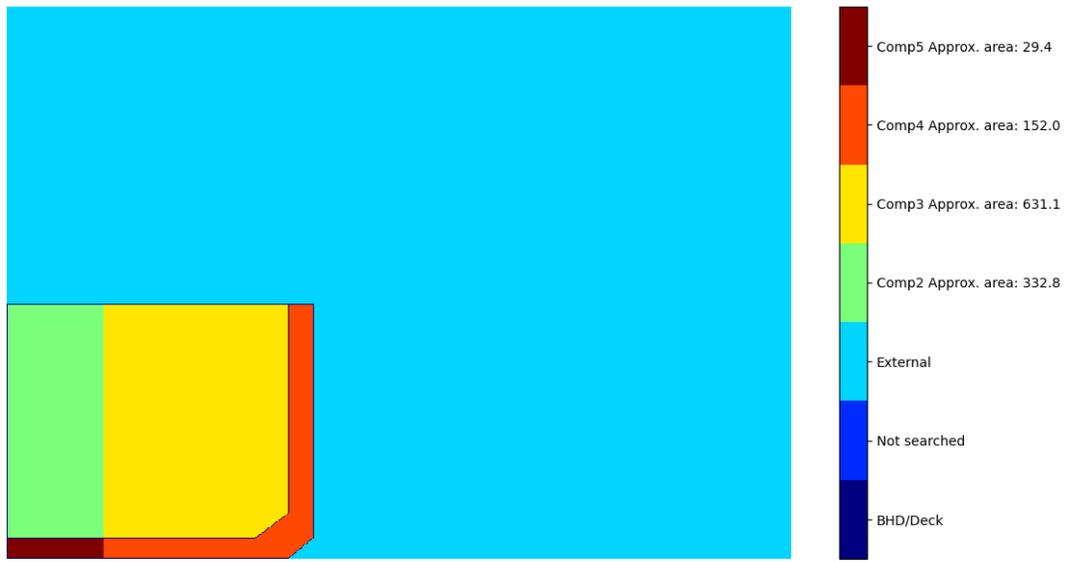
Set compartment properties.

Delete all tanks

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. I is set in the upper right corner as seen next.



Static and dynamic accelerations		line10
Static acceleration [m/s ²]:	<input type="text" value="9.81"/>	Set accelerations
Dyn. acc. loaded [m/s ²]:	<input type="text" value="3.0"/>	
Dyn. acc. ballast [m/s ²]:	<input type="text" value="3.0"/>	

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.

Load properties

1. Dynamic loads
2. Static loads
3. Slamming pressure

Define dynamic loads as a 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**

Hydrostatic loads defined by draft.

Define name of static load: **Create static load**

Define static draft from sea:

Select load condition:

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

3. Created loads are seen below
(scroll if not all is shown.)
DOUBLE CLICK load to see associated lines -

Select to see associated lines: **Delete selected load**

ballast_side	line50
ballast_bottom	line51
loaded_static	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.

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 γ_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 γ_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^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

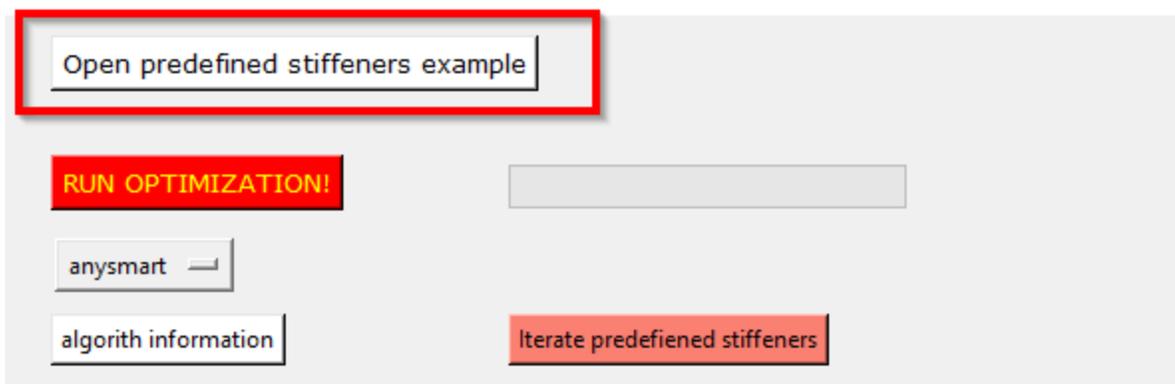
Fatigue results (DNVGL-RP-C203):
Total damage (DFF not included): 0.058 | With DFF = 2.0 --> Damage: 0.117
```

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

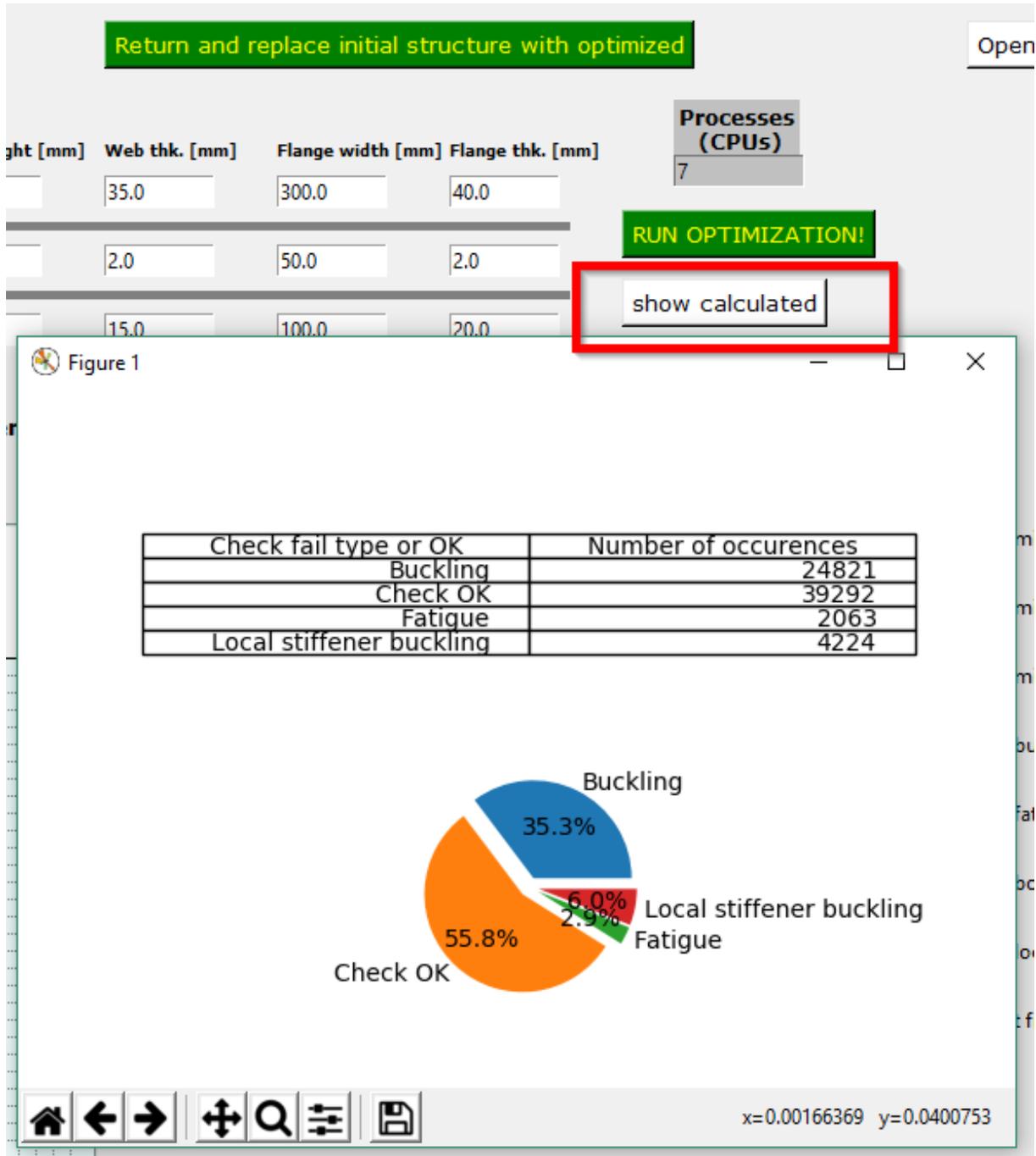
Various checks in the optimization module:

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

-- Structural optimizer for multiple selections --

Accumulated running time: 4.78703465334473 sec

Upper bounds [mm]	Spacing [mm]	Plate thk. [mm]	Web height [mm]	Web thk. [mm]	Flange width [mm]	Flange thk. [mm]
800	50.0	25	500	22	250	30

Iteration delta [mm]: 50.0, 2.0, 50.0, 2.0, 50.0, 2.0

Lower bounds [mm]	Spacing [mm]	Plate thk. [mm]	Web height [mm]	Web thk. [mm]	Flange width [mm]	Flange thk. [mm]
600	10	10	300	10	50	10

Estimated running time for algorithm: 0 seconds

Return and replace with selected optimized structure

Open predefined stiffeners example

algorithm information

Processes (CPUs): 11

Check to harmonize results. Same stiffener and plate dimensions. (defined by largest in opt.)

Initial - Pl: 750.0x15.0 Stf: 400.0x18.0+150.0x20.0
 Weight (per Lg width): 8873

Optimized - Pl: 600.0x14.0 Stf: 350.0x12.0+150.0x10.0
 Weight (per Lg width): 7259

Lateral pressure: 200 kPa

- 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
- Check for local stf. buckling

Mouse left click: select lines to load
 Mouse mid click: show properties for one line
 Mouse right click: clear all selection
 Shift key press: add selected line
 Control key press: remove selected line

NOTE! Select lines you want to return before pressing return button.

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

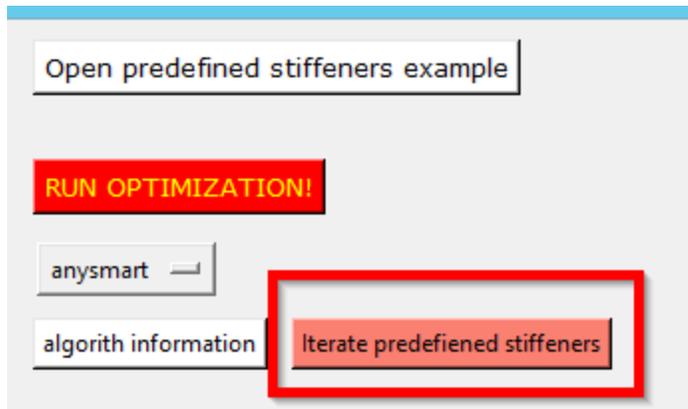
1. Same input on upper bounds, lower bounds and delta.
2. Click all the lines you want to include in the optimization.
3. Run the optimization.
4. Check the properties by **middle clicking** the line you ran.
5. If you are happy return the properties by clicking the top button. Remember to select the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

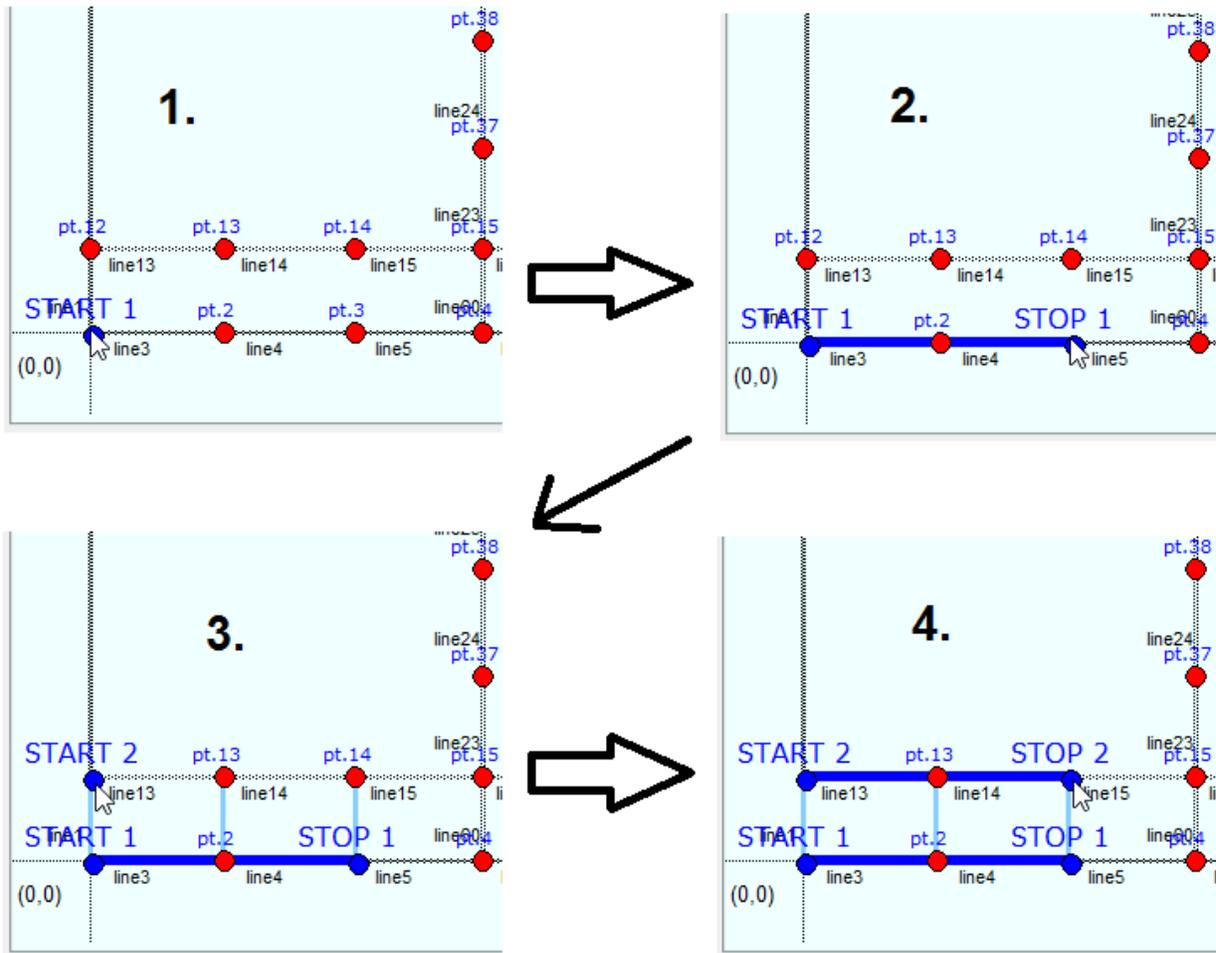
Span optimization

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



The optimization is started as follows.

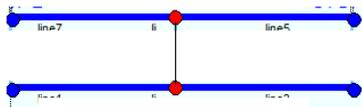
1. Start by clicking as illustrated next:



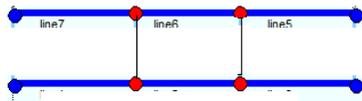
2. Then run optimization.

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.

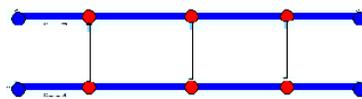
4 plate fields



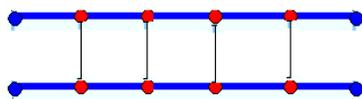
6 plate fields



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

Results are presented as seen next.

RUN OPTIMIZATION!

anysmart

algorithm information

Results seen next. Weight index is tot_weight / max_weight
max_weight is the highest total weight of the checked variations.
Weight index of 1 is the heaviest calculated variation.

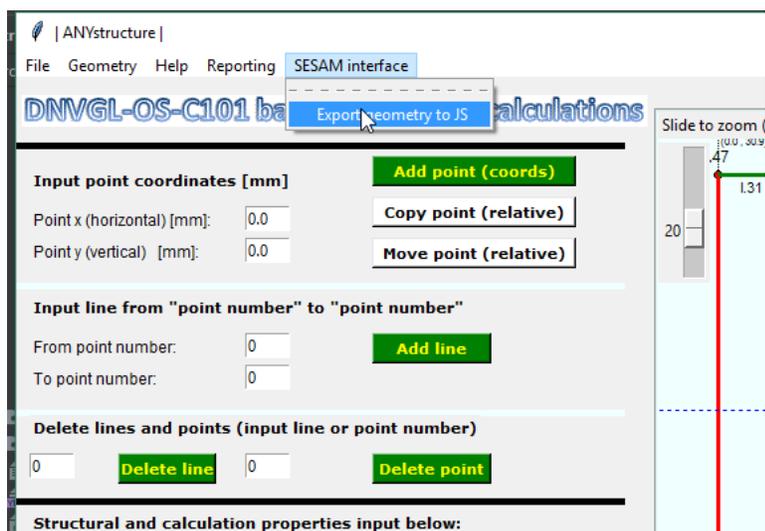
Plate fields	Fields length	Weight index	All OK?

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

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

Export to JS

ANYstructure can export points, lines and section properties to SESAM GenIE. A dialog will request a location to save the JS file. After that you can read the js file into GenIE.



The result is illustrated below:

