

# ANYstructure documentation

File Geometry Help Reporting

### DNVGL-QS-C101 based structural calculations

Slide to zoom (or use mouse wheel) Mouse left click: select line  
Mouse right click: select point

**Input point coordinates [mm]**

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

Point y (vertical) [mm]: 0.0 Copy point (relative)

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 (input line or point number)**

9 Delete line

0 Delete point

**Structural and calculation properties input below:**

span	s	pl_thk	web_h	web_thk	fl_w	fl_thk
3.8	700.0	18.0	400.0	12.0	200.0	18.0
[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	df type	pressure side	
100.0	100.0	100.7	5.0	IT	p	

Material yield [MPa]: 355.0

Select structure type: BOTTOM

Internal pressure from comp.

Add structure to line

**SELECTED: line9**

Applied compartments: Compartment 4

Applied static/dynamic loads: ballast\_bottom

**Static and dynamic accelerations** line9

Static acceleration [m/s<sup>2</sup>]: 9.81 Get accelerations

Dyn. acc. loaded [m/s<sup>2</sup>]: 3.0

Dyn. acc. ballast [m/s<sup>2</sup>]: 3.0

**Optimize selected line/structure (right click line):**

OPTIMIZE MultiOpt SPAN

Combination for line (select line). Change with slider:

OS-C101 Table 1: DNV a) 2: DNV b) 3: TankTest

Name	Stat LP	Dyn LP	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) ballast; (DNV b) tank test; manual)  
 Note that ch. 4.3.7 and 4.3.8 is accounted for.

DNV a [Pa]: 126661.236396      DNV b [Pa]: 179008.231728  
 TT [Pa]: 1337077      Manual [Pa]: 0.0

**Section moduluses: Wey1: 4.7000E+06 [mm<sup>3</sup>]; Wey2: 1.9600E+06 [mm<sup>3</sup>]**

Minimum section modulus: 1.9421E+06 [mm<sup>3</sup>]

**Shear area: 5.2320E+03 [mm<sup>2</sup>]**

Minimum shear area: 3.7877E+03 [mm<sup>2</sup>]

**Plate thickness: 18.0 [mm]**

Minimum plate thickness: 14.7 [mm]

**Buckling results DNV-RP-C201 (z\* optimized):**  
 [eq. 7.19: 0.83] [eq. 7.50: 0.92] [eq. 7.53: -0.13] [7.52: 0.62] [eq. 7.53: 0.92] [z\*: 0.14]

**Fatigue results (DNVGL RP-C203):**

Total damage (DFF not included): 0.03 | With DFF = 2.0 --> Damage: 0.06

Find compartments External pressures

Display current compartments

Comp. no.:

2	Tank content:		<span style="float: right;">Set compartment properties.</span>
3	Tank density:	1025 [kg/m <sup>3</sup> ]	
4	Overpressure:	23000 [Pa]	<span style="float: right;">Delete all tanks</span>
5	Max elevation:	0.0	
	Min elevation:	0.0	
	Acceleration [m/s <sup>2</sup> ]:		

April 2019

Version 0.6.x

# Table of contents

<b>Table of contents</b>	<b>2</b>
<b>Modelling</b>	<b>3</b>
<b>Assigning properties</b>	<b>4</b>
<b>Define tanks</b>	<b>4</b>
<b>Define external pressures</b>	<b>5</b>
<b>Load combinations</b>	<b>6</b>
<b>Optimization</b>	<b>7</b>
Optimization iteration by predefined stiffeners	7
Single optimization	8
Multiple optimization	9
Span optimization	10

# 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 (input line or point number)</b>		
<input type="text" value="0"/>	<b>Delete line</b>	<input type="text" value="0"/>
		<b>Delete point</b>

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

## 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.

**Define plate and beam properties.**

spen	s	pl_thk	web_h	web_thk	fl_w	fl_thk
4.0	750.0	18.0	350.0	12.0	150.0	20.0
[m]	[mm]	[mm]	[mm]	[mm]	[mm]	[mm]
kpp	kps	km1	km2	k3		
1.0	1.0	12.0	24.0	12.0		
[mm]	[mm]	[mm]	[mm]	[mm]		

**FLS** Material yield [MPa] 355.0  
Select structure type -> BOTTOM  
Internal pressure from comp. Add structure to line

**Define calculation properties.**

**Define fatigue properties.**

**Define structure properties here --**

Stiffener type: T

Spacing: 750.0 [mm]

Plate thk.: 18.0 [mm]

Web height: 350.0 [mm]

Web thk.: 12.0 [mm]

Flange width: 150.0 [mm]

Flange thk.: 20.0 [mm]

Plate: 750.0x18.0  
Web: 350.0x12.0  
Flange: 150.0x20.0

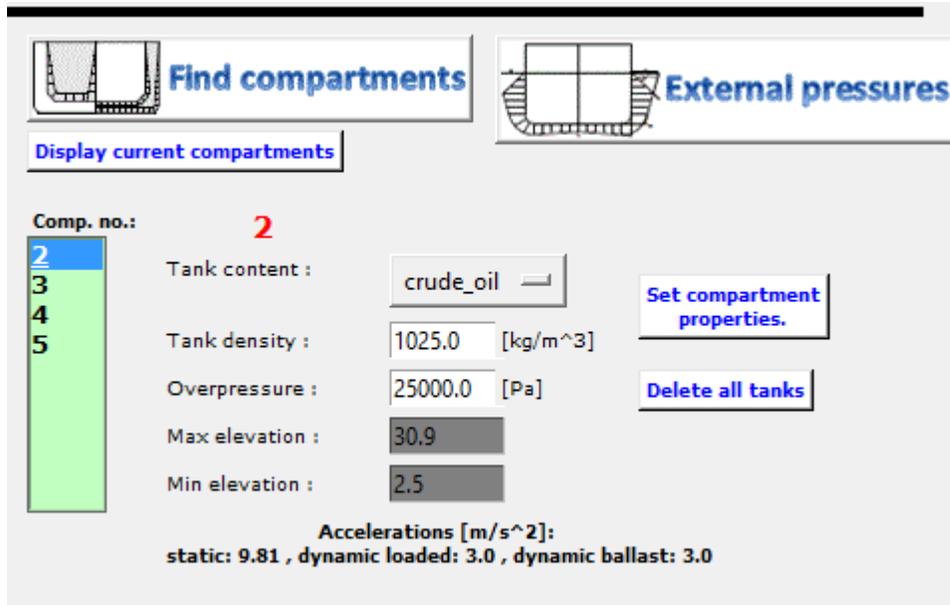
Girder length (Lg) 10

Save and return structure

## 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.



## 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.

**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: **Press this to: Save loads and close the load window.**

**3. Slamming pressure**

Load name: slamming  
 Pressure [Pa]: 0.0 **Create slamming load**

**3. Created loads are seen below (double click to select):**

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

ballast\_side  
 ballast\_bottom  
 loaded\_static  
 ballast\_static  
 slamming  
 loaded\_bottom

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

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

line1 line2 line3 line4 line5 line6 line7 line8 line9 line10 line11 line12 line13 line14 line15 line16 line17 line18 line19 line20 line21 line22 line23 line24 line25 line26 line27 line28 line29 line30 line31 line32 line33 line34 line35 line36 line37 line38 line39 line40 line41 line42 line43 line44 line45 line46 line47 line48 line49 line50 line51 line52 line53 line54 line55 line56 line57 line58 line59 line60 line61 line62 line63 line64 line65 line66 line67 line68 line69 line70 line71 line72 line73

## 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.

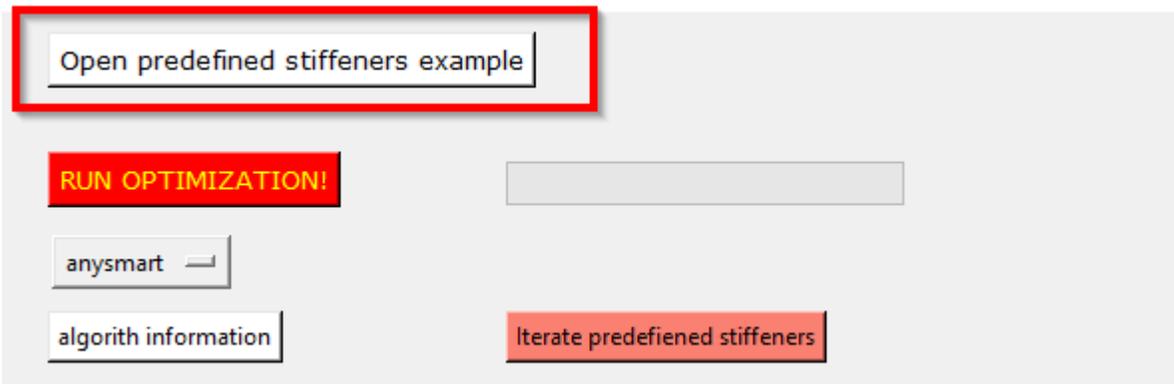
3. You can deselect a load by manually inputting load factor to 0 or deselect include.

## Optimization

### Optimization iteration by predefined stiffeners

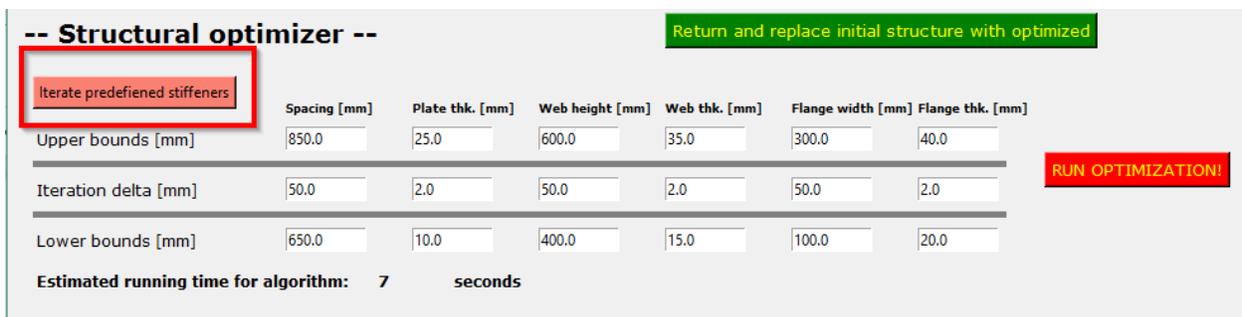
From 0.5 you can iterate by a defined set of stiffeners. Press the button marked below. Open a csv (or json) file. Then start your iterations. The only other input is the stiffener spacing and plate thickness.

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.



## 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

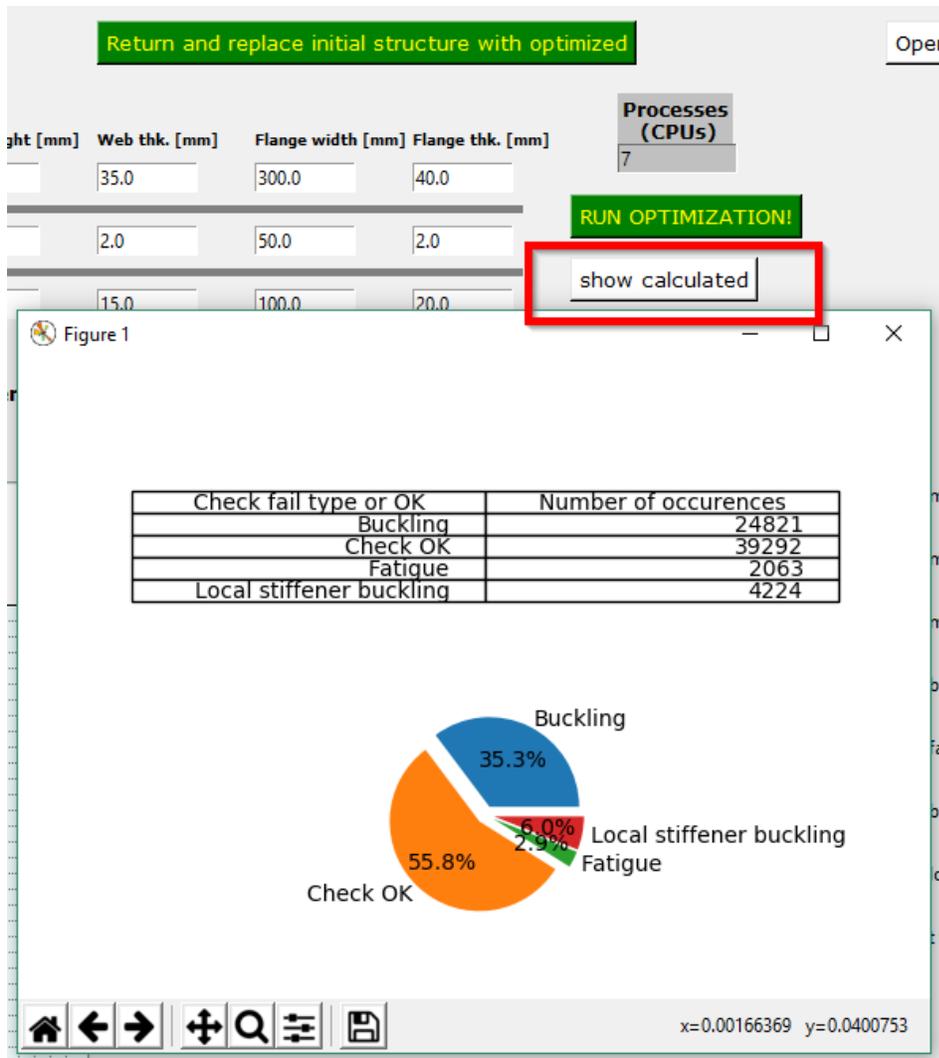
Various checks in the optimization module:

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).



## Multiple optimization

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

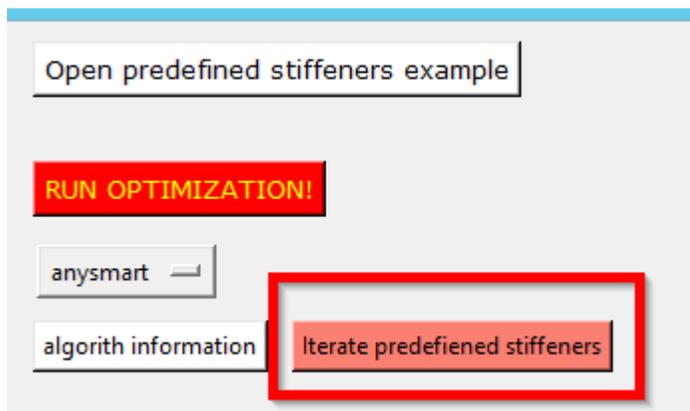
1. Same input on upper bounds, lower bounds and delta.
2. Click all the lines you want to include in the optimization.
3. Run the optimization.
4. Check the properties by **middle clicking** the line you ran.
5. If you are happy return the properties by clicking the top button

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

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

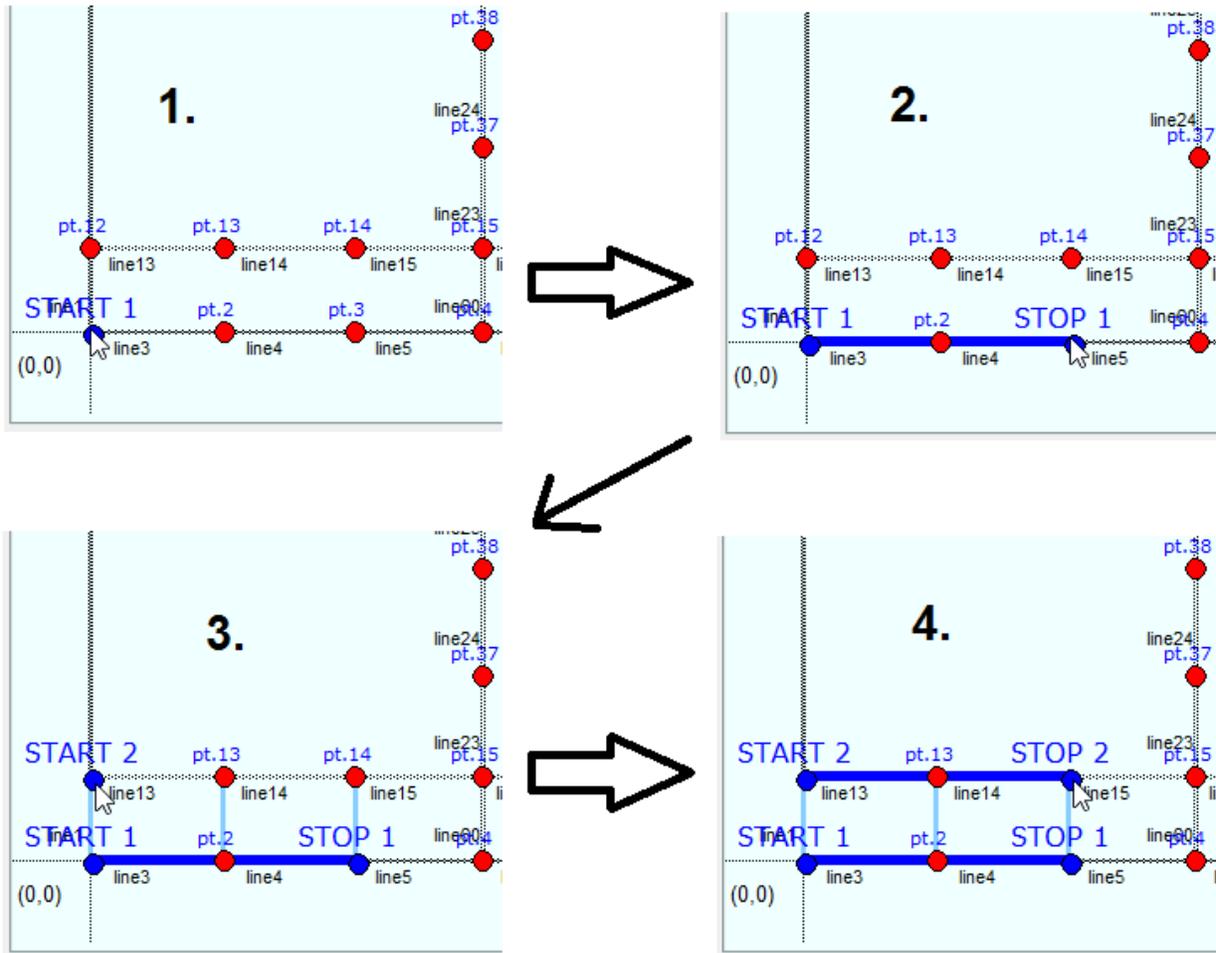
## Span optimization

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



The optimization is started as follows.

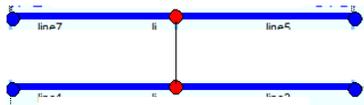
1. Start by clicking as illustrated next:



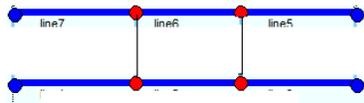
## 2. Then run optimization.

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.

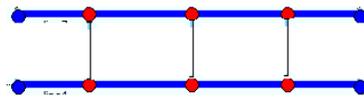
### 4 plate fields



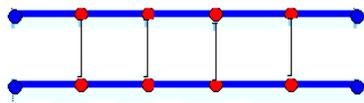
### 6 plate fields



### 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.

When the analysis has been runned you should save your results. Just specify a file name in the save file dialog.

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.

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