

# ANYstructure documentation

ANYstructure | File Geometry Help Reporting

DNVGL-QS-C101 based structural calculations

Slide to zoom (or use mouse wheel)

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

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

Delete lines and points (input line or point number):  
Delete line: 0  
Delete point: 0

Structural and calculation properties input below:

FLS  
Material yield [MPa]: 355.0  
Select structure type: BOTTOM  
Buttons: Show structure types, Add structure to line

Find compartments  
External pressures

Display current compartments

Comp. no.: 2  
Tank content: 1025 [kg/m³]  
Tank density: 1025 [kg/m³]  
Overpressure: 23000 [Pa]  
Max elevation: 0.0  
Min elevation: 0.0  
Acceleration [m/s²]:

Buttons: Set compartment properties, Delete all tanks

Static and dynamic accelerations  
Static acceleration [m/s²]: 9.81  
Dyn. acc. loaded [m/s²]: 3.0  
Dyn. acc. ballast [m/s²]: 3.0  
Buttons: Set accelerations

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: ballast\_bottom  
loaded\_static  
ballast\_static  
loaded\_bottom  
Compartment4  
Manual (pressure/LF): 0.0 1.0

Pressure 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): 126661, 236436  
DNV b (Pa): 147608, 231728  
TT (Pa): 1335707  
Manual (Pa): 0.0

Section moduluses: Wey1: 4.7000E+06 [mm³], Wey2: 1.9600E+06 [mm³]  
Minimum section modulus: 1.9421E+06 [mm³]  
Shear area: 5.2320E+03 [mm²]  
Minimum shear area: 3.7877E+03 [mm²]  
Plate thickness: 18.0 [mm]  
Minimum plate thickness: 14.7 [mm]  
Buckling results DNV-RP-C201 (a\* optimized):  
[eq 7.19: 0.83] [eq 7.50: 0.92] [eq 7.51: -0.13] [7.52: 0.62] [eq 7.53: 0.92] [a\*: 0.14]  
Fatigue results (DNVGL-CP-C203):  
Total damage (DFF not included): 0.03 | With DFF = 2.0 --> Damage: 0.06

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

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	10
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

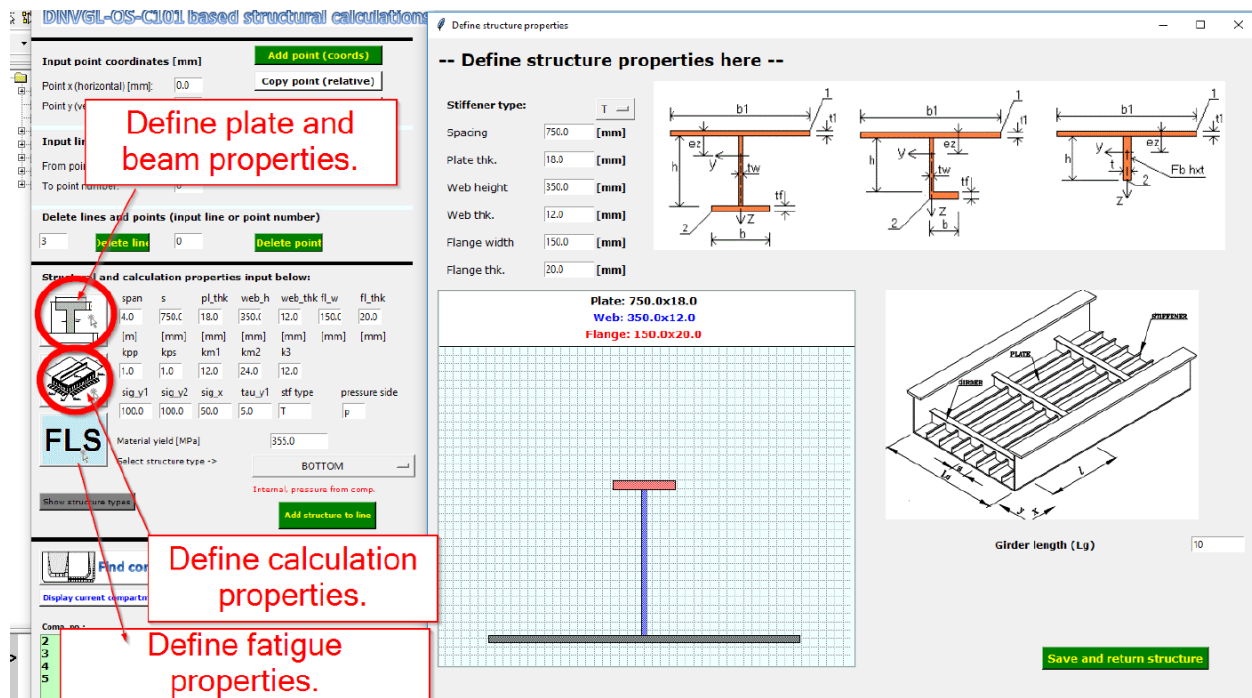
CTRL-C Copy a selected point

CTRL-M Move a selected point

CTRL-Q New line between two selected points

CTRL-S Assign properties to a selected line

**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 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.**

The screenshot shows a software interface with two main tabs: "Find compartments" and "External pressures". The "Find compartments" tab is active, showing a list of compartments (2, 3, 4, 5) on the left. Compartment 2 is selected, and its properties are displayed on the right. The properties include Tank content (crude\_oil), Tank density (1025.0 [kg/m^3]), Overpressure (25000.0 [Pa]), Max elevation (30.9), and Min elevation (2.5). There are buttons for "Set compartment properties" and "Delete all tanks". At the bottom, accelerations are listed: static: 9.81, dynamic loaded: 3.0, dynamic ballast: 3.0.

Find compartments

External pressures

Display current compartments

Comp. no.: **2**

2  
3  
4  
5

Tank content : crude\_oil

Tank density : 1025.0 [kg/m<sup>3</sup>]

Overpressure : 25000.0 [Pa]

Max elevation : 30.9

Min elevation : 2.5

Set compartment properties.

Delete all tanks

Accelerations [m/s<sup>2</sup>]:  
static: 9.81 , dynamic loaded: 3.0 , dynamic ballast: 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**

**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:  **Create static load**

Define static draft from sea:

Select load condition:

**3. Slamming pressure**

Load name:

Pressure [Pa]:  **Create slamming load**

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

**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  
 Drag 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

## 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.

Open predefined stiffeners example

RUN OPTIMIZATION!

anysmart

algorithm information

Iterate predefined stiffeners

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

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

-- Structural optimizer --

Return and replace initial structure with optimized

Iterate predefined stiffeners

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

Estimated running time for algorithm: 7 seconds

RUN OPTIMIZATION!

## Single optimization

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

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

Various checks in the optimization module:

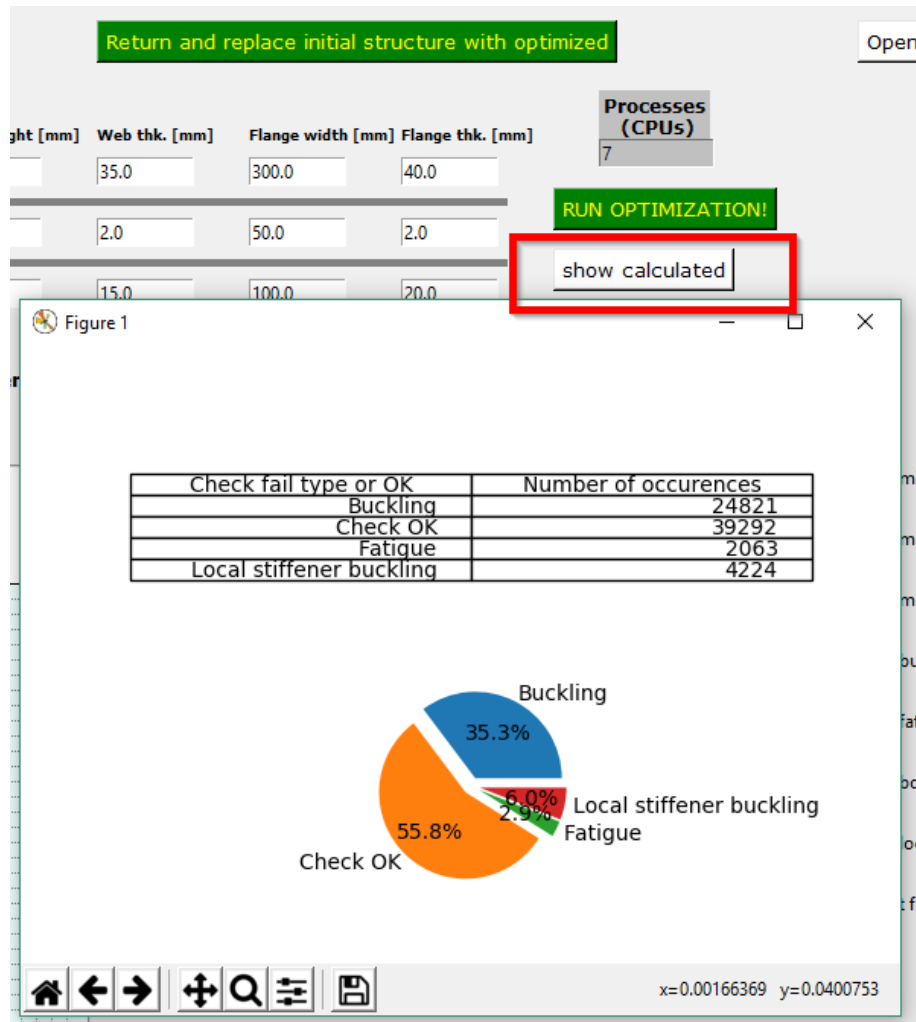
You can select the checks to be performed.

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

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

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





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.

	A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P	Q	R	S
1	Is OK	Check info	pl b	pl thk	web h	web thk	fl b	fl thk	span	girder width	stiffener ty	uf weight	uf sec mod	uf loc stf bu	uf buckling	uf min pl	uf shear	uf fatigue	uf slamming
2	FALSE	Section moi	0.65	0.01	0.06	0.004	0.0165	0.008919	4	10 L		0	223.1465	0	0	0	0	0	0
3	FALSE	Section moi	0.65	0.012	0.06	0.004	0.0165	0.008919	4	10 L		0	117.4682	0	0	0	0	0	0
4	FALSE	Section moi	0.65	0.014	0.06	0.004	0.0165	0.008919	4	10 L		0	83.38551	0	0	0	0	0	0
5	FALSE	Section moi	0.65	0.016	0.06	0.004	0.0165	0.008919	4	10 L		0	65.39368	0	0	0	0	0	0
6	FALSE	Section moi	0.65	0.018	0.06	0.004	0.0165	0.008919	4	10 L		0	53.72824	0	0	0	0	0	0
7	FALSE	Section moi	0.65	0.02	0.06	0.004	0.0165	0.008919	4	10 L		0	45.29623	0	0	0	0	0	0
8	FALSE	Section moi	0.65	0.022	0.06	0.004	0.0165	0.008919	4	10 L		0	38.75803	0	0	0	0	0	0
9	FALSE	Weight filte	0.65	0.024	0.06	0.004	0.0165	0.008919	4	10 L		1.016862	0	0	0	0	0	0	0
10	FALSE	Section moi	0.7	0.01	0.06	0.004	0.0165	0.008919	4	10 L		0	243.9762	0	0	0	0	0	0
11	FALSE	Section moi	0.7	0.012	0.06	0.004	0.0165	0.008919	4	10 L		0	126.6436	0	0	0	0	0	0
12	FALSE	Section moi	0.7	0.014	0.06	0.004	0.0165	0.008919	4	10 L		0	89.35638	0	0	0	0	0	0
13	FALSE	Section moi	0.7	0.016	0.06	0.004	0.0165	0.008919	4	10 L		0	69.74879	0	0	0	0	0	0
14	FALSE	Section moi	0.7	0.018	0.06	0.004	0.0165	0.008919	4	10 L		0	57.05856	0	0	0	0	0	0
15	FALSE	Section moi	0.7	0.02	0.06	0.004	0.0165	0.008919	4	10 L		0	47.90213	0	0	0	0	0	0
16	FALSE	Section moi	0.7	0.022	0.06	0.004	0.0165	0.008919	4	10 L		0	40.82009	0	0	0	0	0	0
17	FALSE	Weight filte	0.7	0.024	0.06	0.004	0.0165	0.008919	4	10 L		1.01576	0	0	0	0	0	0	0

## Multiple optimization

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

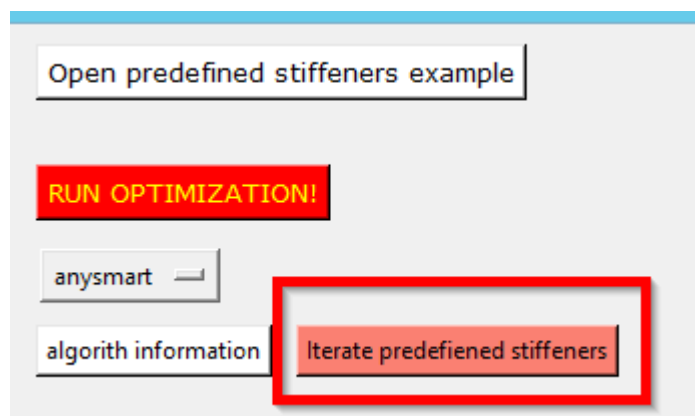
1. Same input on upper bounds, lower bounds and delta.
2. Click all the lines you want to include in the optimization.
3. Run the optimization.
4. Check the properties by **middle clicking** the line you ran.
5. If you are happy return the properties by clicking the top button

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

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

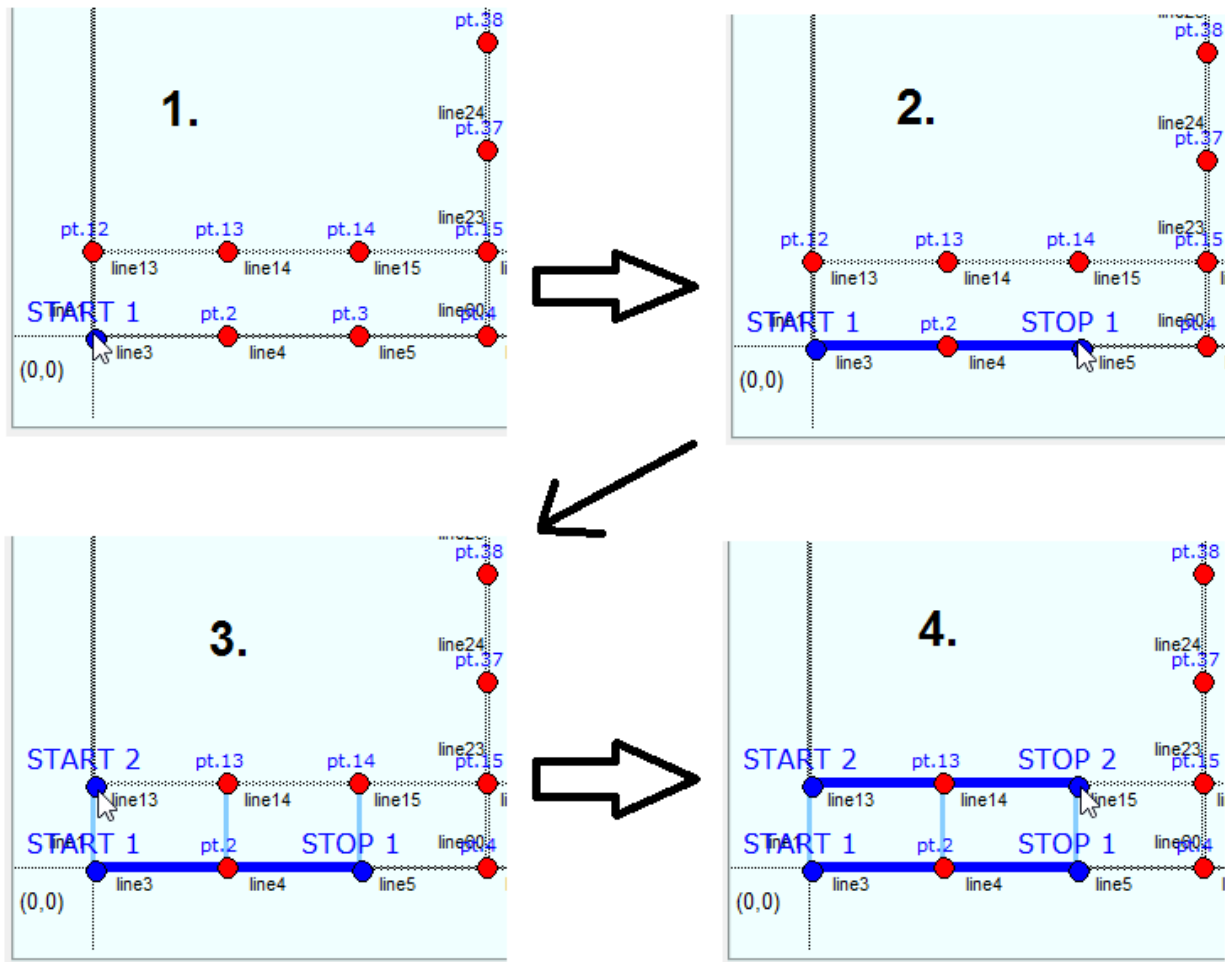
## Span optimization

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



The optimization is started as follows.

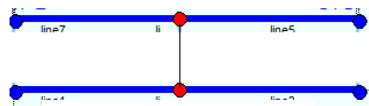
1. Start by clicking as illustrated next:



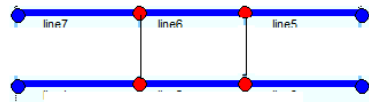
2. Then run optimization.

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.

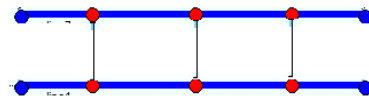
#### 4 plate fields



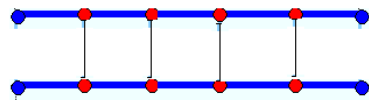
#### 6 plate fields



#### 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 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.

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