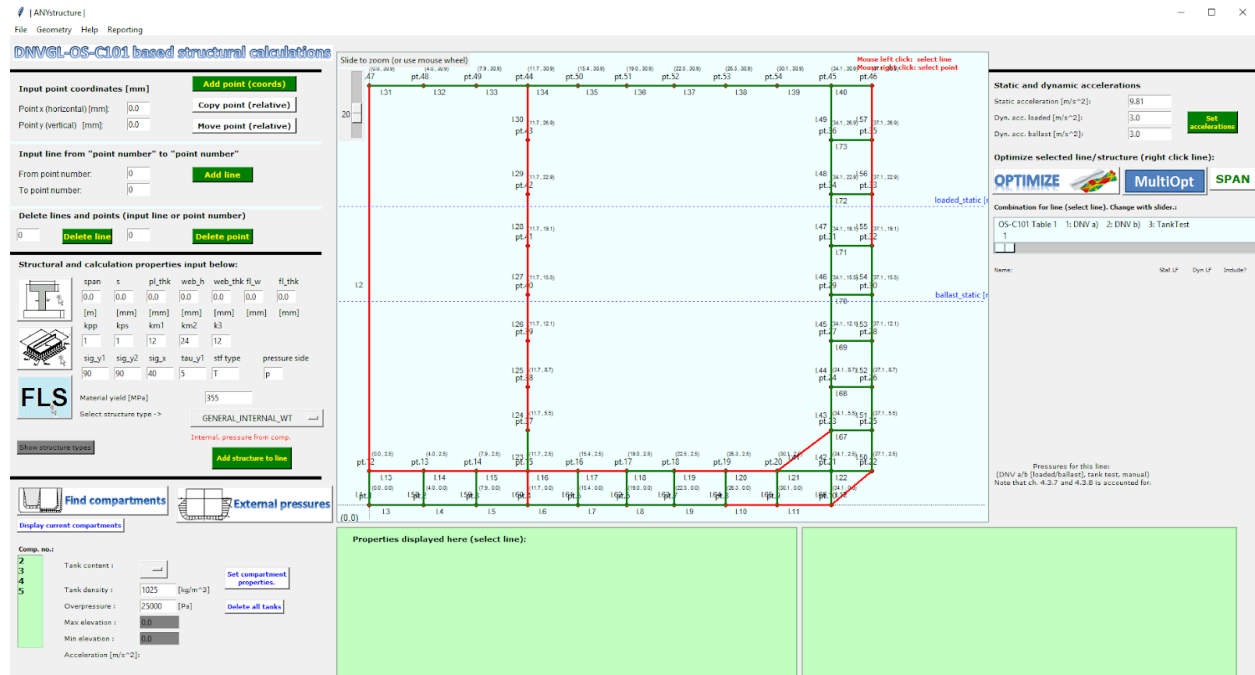


# ANYstructure documentation



March 2019

Version 0.5.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	7
Multiple optimization	7
Span optimization	8

# 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.

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

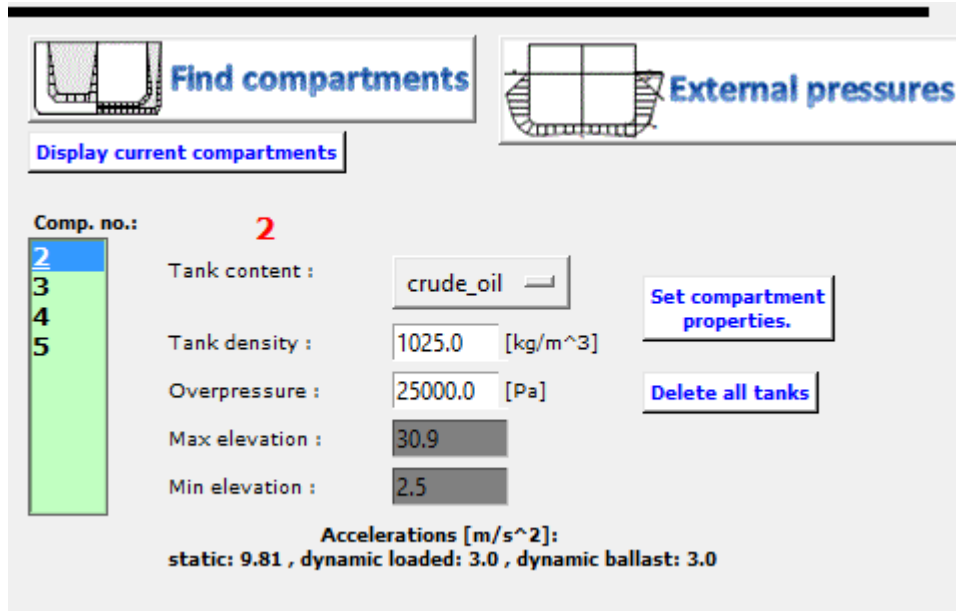
Girder length (Lg) 10

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.



**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:

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.

**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

House left click: select lines to loads  
House right click: clear all selection  
Shift key: add selected line  
Control key press: remove selected line

## 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**.

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.

-- 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.

### Multiple optimization

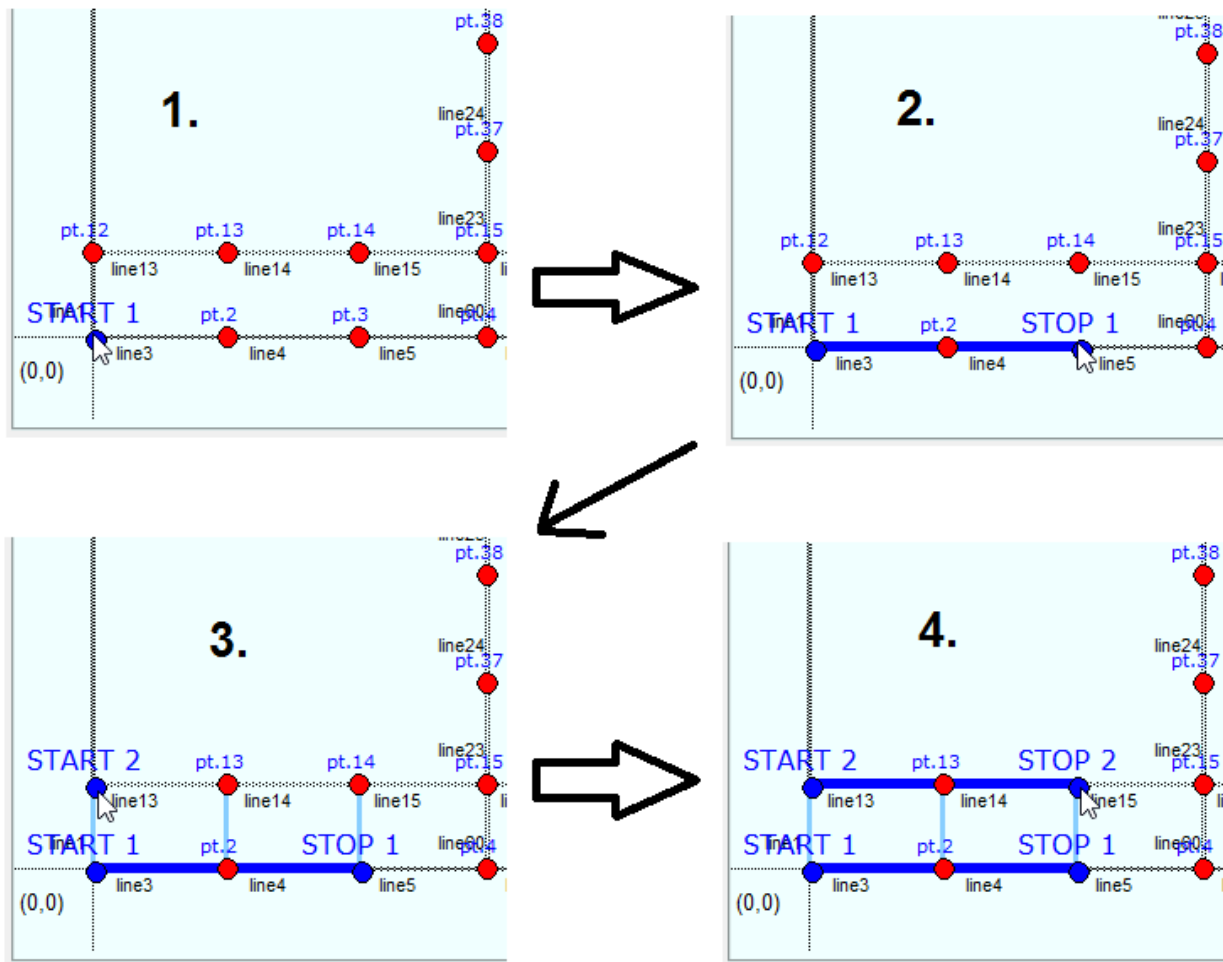
Multiple optimization is done by clicking the “MultiOpt” button.

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 right clicking the line.
5. If you are happy return the properties by clicking the top button

## Span optimization

**NOTE:** The span optimization is computationally heavy.

1. Start by clicking as illustrated next:

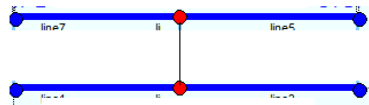




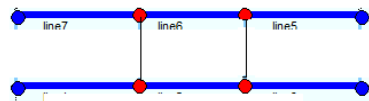
**2. Then run optimization.**

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.

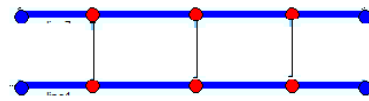
**4 plate fields**



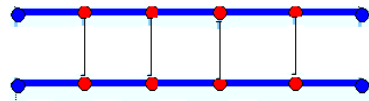
**6 plate fields**



**8 plate fields**



**10 plate fields**



**Results are presented as seen next.**

RUN OPTIMIZATION!

anysmart

algorithm information

Results seen next. Weight index is  $\text{tot\_weight} / \text{max\_weight}$ .  
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.