

ANYstructure documentation

The screenshot displays the ANYstructure software interface, which is used for structural analysis and documentation. The interface is divided into several main sections:

- Top Left: File and Settings**
 - Menu: File Geometry Help Reporting
 - Version: DNVGL-OS-C101 based structural calculations
- Left Panel: Input and Properties**
 - Input point coordinates [mm]**: Fields for Point x (horizontal) [mm] and Point y (vertical) [mm], both set to 0.0. Buttons: Add point (coords), Copy point (relative), Move point (relative).
 - Input line from "point number" to "point number"**: Fields for From point number and To point number, both set to 0. Button: Add line.
 - Delete lines and points (input line or point number)**: Fields for line and point numbers, both set to 0. Buttons: Delete line, Delete point.
 - Structural and calculation properties input below:**
 - Material yield [MPa]: 355
 - Material: GENERAL_INTERNAL_WT
 - Buttons: Show structure types, Add structure to line.
 - Find compartments** and **External pressures** sections.
- Center: 3D Model**
 - Slide to zoom (or use mouse wheel)
 - Mouse left click: select line; Mouse right click: select point
 - Model shows a vessel structure with points labeled pt.1 through pt.20 and various dimensions.
 - Labels: loaded_static, ballast_static
- Right Panel: 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: DNV a) 1: DNV a) 2: DNV b) 3: TankTest
 - Table with 1 column and 1 row.
 - Buttons: Stat LP, Dyn LP, Include?
 - 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.
- Bottom Left: Comp. no.**
 - Buttons: 2, 3, 4, 5
 - Tank content: [field]
 - Buttons: Set compartment properties, Delete all tanks
 - Tank density: 1025 [kg/m³]
 - Overpressure: 25000 [Pa]
 - Max elevation: 0.0
 - Min elevation: 0.0
 - Acceleration [m/s²]: [field]
- Bottom Center: Properties displayed here (select line):** A large green area for displaying properties.

March 2019

Version 0.4

Table of contents

Table of contents	2
Modelling	3
Assigning properties	4
Define tanks	4
Define external pressures	5
Load combinations	6
Optimization	7
Single optimization	7
Multiple optimization	7
Span optimization	7

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

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]		
sig_y1	sig_y2	sig_x	tau_y1	stf type	pressure side	
100.0	100.0	50.0	5.0	T	p	
[MPa]	[MPa]	[MPa]	[MPa]			

Define calculation properties.

Define fatigue properties.

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

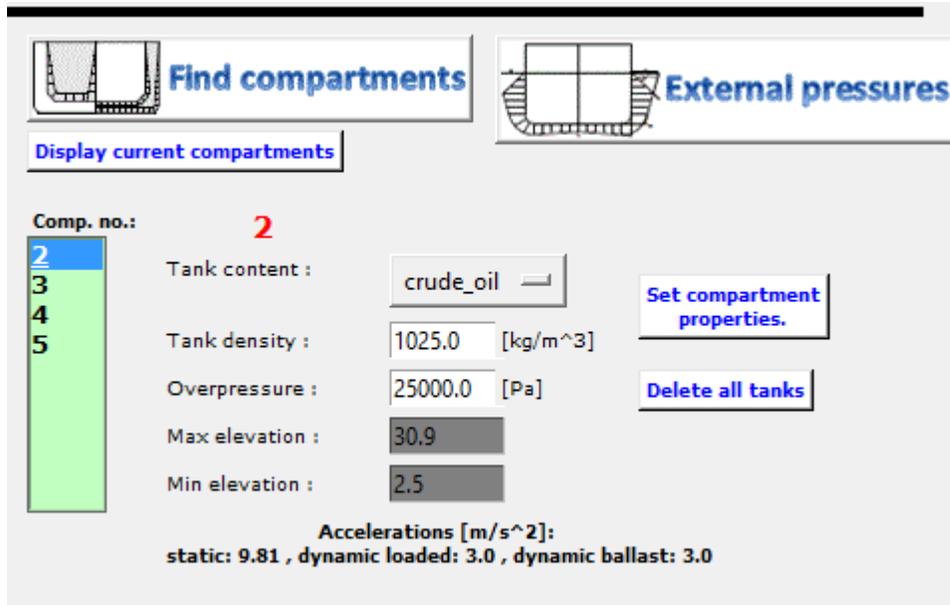
Girder length (Lg) 10

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: **ballast**

3. Slamming pressure

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

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

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

Select to see associated lines:

ballast_side	line50
ballast_bottom	line51
loaded_static	line52
ballast_static	line53
slamming	line54
loaded_bottom	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 ----->

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

ballast_side

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

(0,0)

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

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

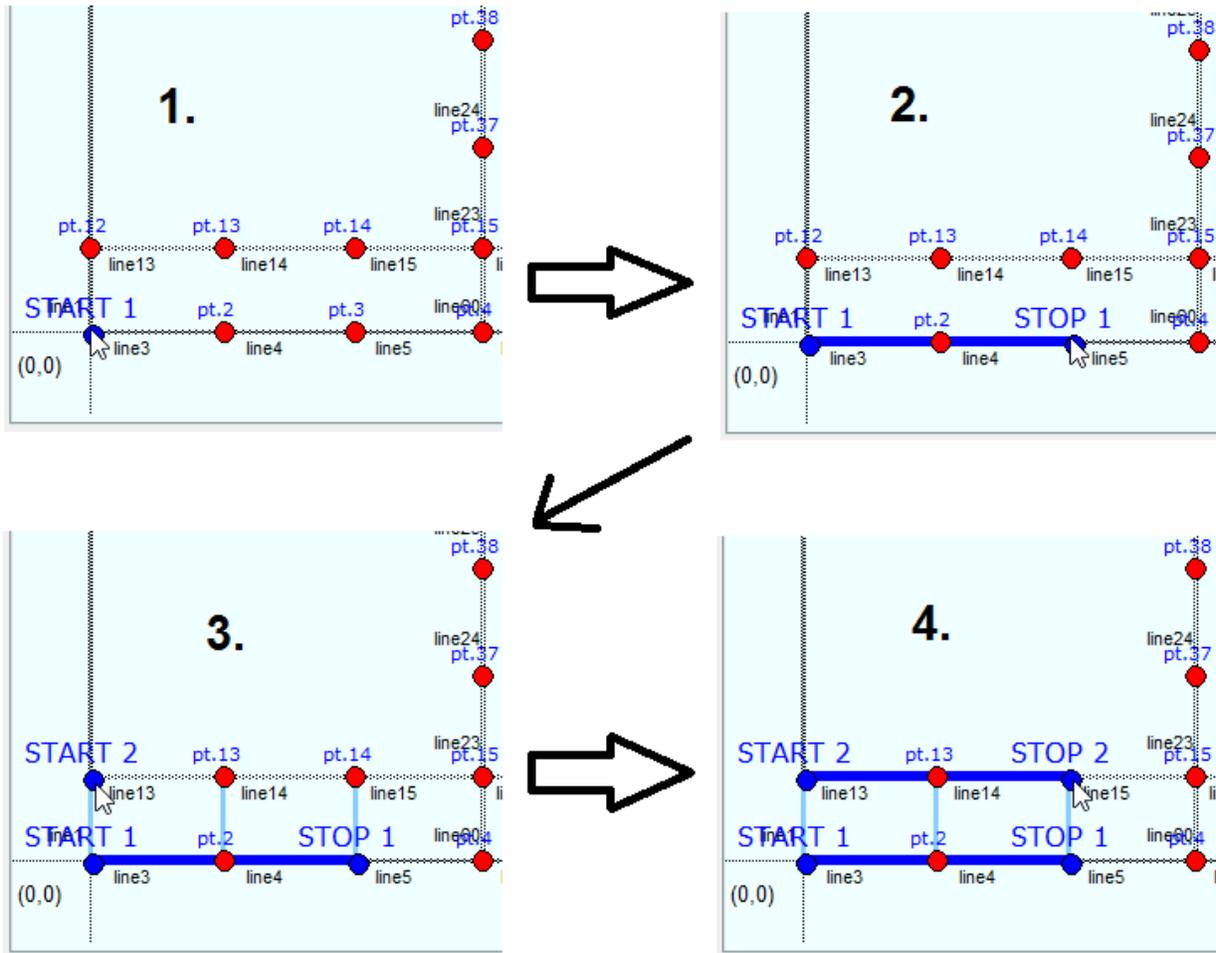
Multiple optimization is done by clicking the “MultiOpt” button.

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.

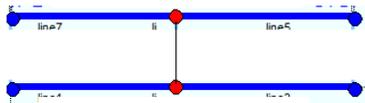
1. Start by clicking as illustrated next:



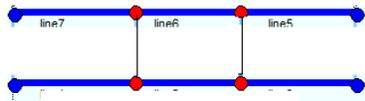
2. Then run optimization.

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.

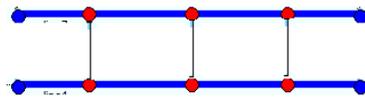
4 plate fields



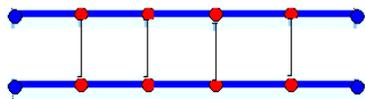
6 plate fields



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.