ANYstructure

Documentation

2021

Version 2.X

Table of contents

Introduction

ANYstructure is a free structural optimization tool. It can be used for multiple purposes. The software can be downloaded various ways:

For python users
PIP install ANYstructure

For windows version
Download at https://github.com/audunarn/ANYstructure/releases or https://sourceforge.net/projects/anystructure/

The code is located on github and is open source (https://github.com/audunarn/ANYstructure)

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**

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]:	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):	43	Delete line
Point number (right click):	0	Delete point

Speed up your modelling <u>significantly</u> 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.

Up and down arrow to change current point.

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".
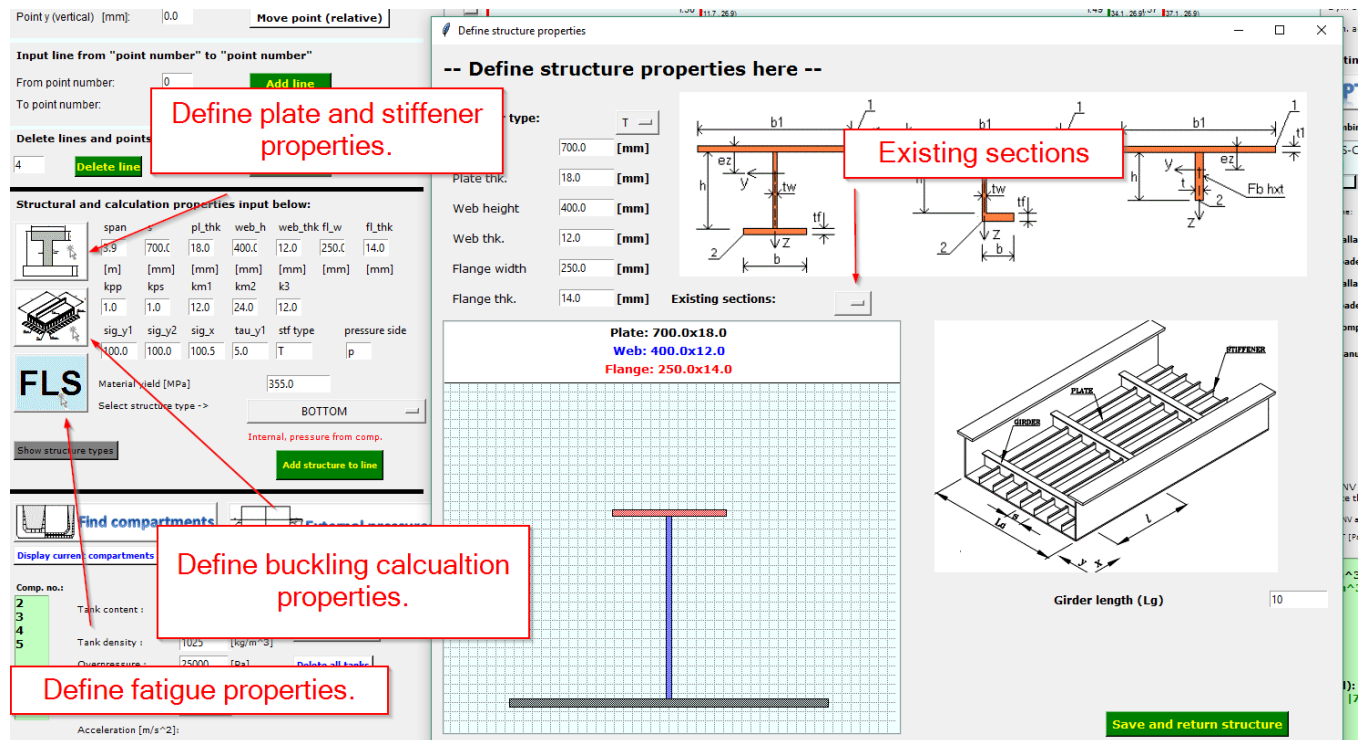
By default z* is ticked. This affects the buckling results and will generally give lower utilization than using z* = 0. See description below.

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.

Plate field span: 3.8 meters
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 varible_   x - horizontal
Plate fixation paramter,kpp: 1.0
Stf. fixation paramter,kps: 1.0
Global stress, sig_y1/sig_y2: 100.0/100.0 MPa
Global stress, sig_x: 100.7 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: line9

Applied compartments: Compartment 4

Applied static/dynamic loads: ballast_bottom
                                         loaded_static
                                         ballast_static
                                         loaded_bottom
                                         fls_ballast

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.

Ather 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

Display current compartments

Tank density : 1000 [kg/m^3]

Overpressure : 25000.0 [Pa]

Set compartment properties.

Max elevation : 30.9

Min elevation : 2.5

Delete all tanks

Accelerations [m/s^2]:
static: 9.81 , dynamic loaded: 3.0 , dynamic ballast: 3.0

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.
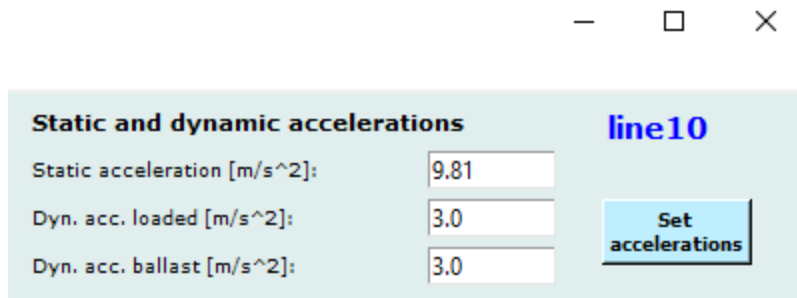
Figure 1

Compartments returned from search operation displayed below

Comp5 Approx. area: 29.4

Comp4 Approx. area: 152.0

Comp3 Approx. area: 631.1

Comp2 Approx. area: 332.8

External

Not searched
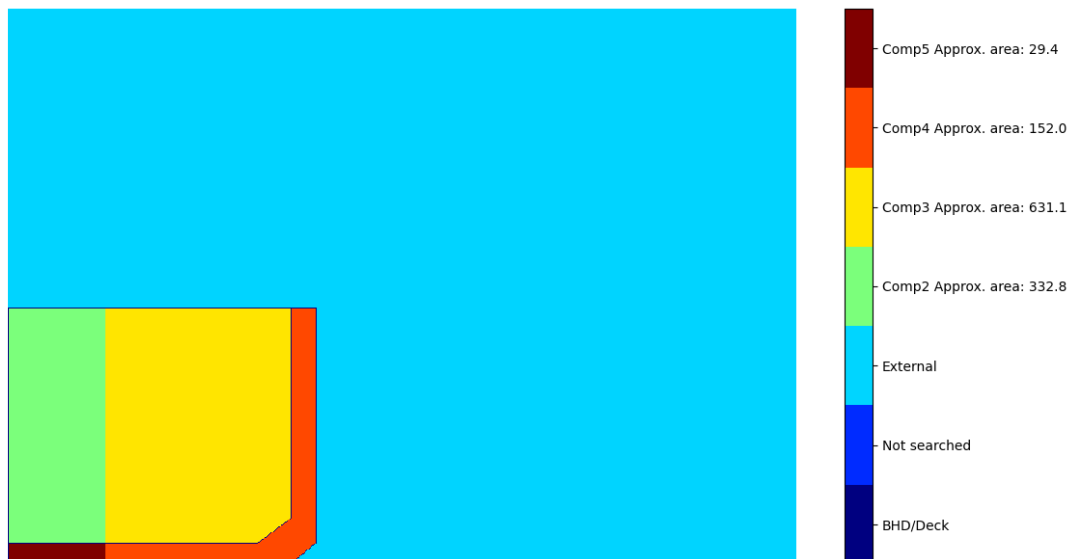
BHD/Deck

*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]:   9.81

Dyn. acc. loaded [m/s^2]:   3.0   **Set accelerations**

Dyn. acc. ballast [m/s^2]:   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 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.**

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.

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

1

me:	Stat LF	Dyn LF	Include?
static_22m	1.3	0	☑
static_15m	1.3	0	☑
static_8m_tt	0	0	☐
loaded_bottom	0	0.7	☑
ballast_bottom	0	0.7	☑
Compartment2	1.2	0.7	☑
Manual (pressure/LF)	0	1	☑

Pressures for this line:
DNV a/b [loaded/ballast], tank test, manual)
ote 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 facors   **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"	1.3
Condition a) - Static load factor well defined loads	1.2
Condition a) - Dynamic load factor	0.7
Condition b) - Static load factor "unknown loads"	1
Condition b) - Static load factor well defined loads	1
Condition b) - Dynamic load factor	1.3
Tank test) - Static load factor "unknown loads"	1
Tank test) - Static load factor well defined loads	1
Tank test) - Dynamic load factor	0

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 γ_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 data

Loads

Pressure magnitude can be reviewed by using color coding. The highest total pressure used in calculations is shown.
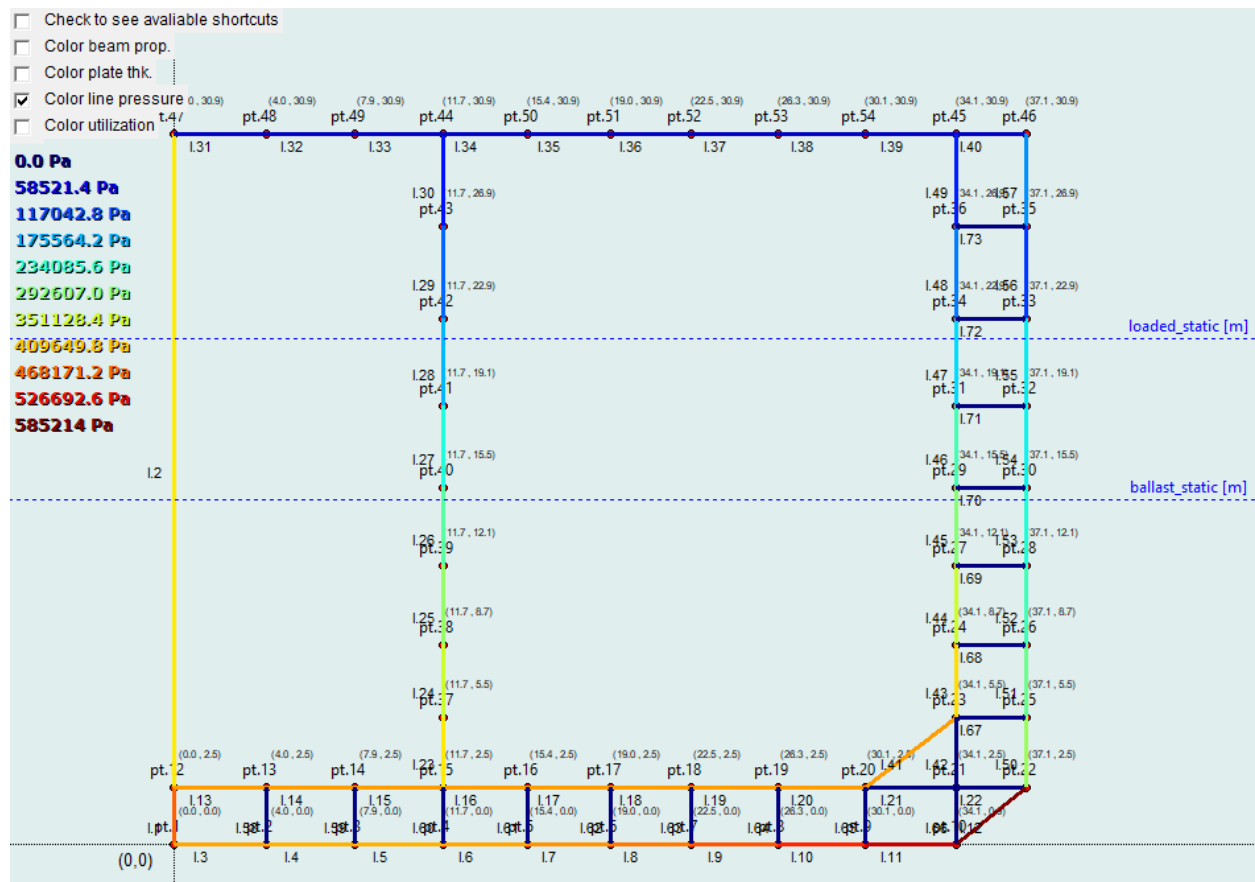
Load calculations and results can be reviewed by clicking the "Load info" button. An example is seen in the next illustration.

Load info for line9   ✕

(i) 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*310707.22500000003 + 25000.0*1.3 =
405348.67000000004
comp4 - dynamic: 1*0.7*95017.50000000001 + 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*310707.22500000003 + 25000.0*1.3 =
343207.22500000003
comp4 - dynamic: 1*1.3*95017.50000000001 + 25000.0*0 =
123522.75000000003

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]

Thickness and beam properties

Line plate thicknesses and beam properties can be reviewed using color coding. Plate thicknesses are exemplified next.
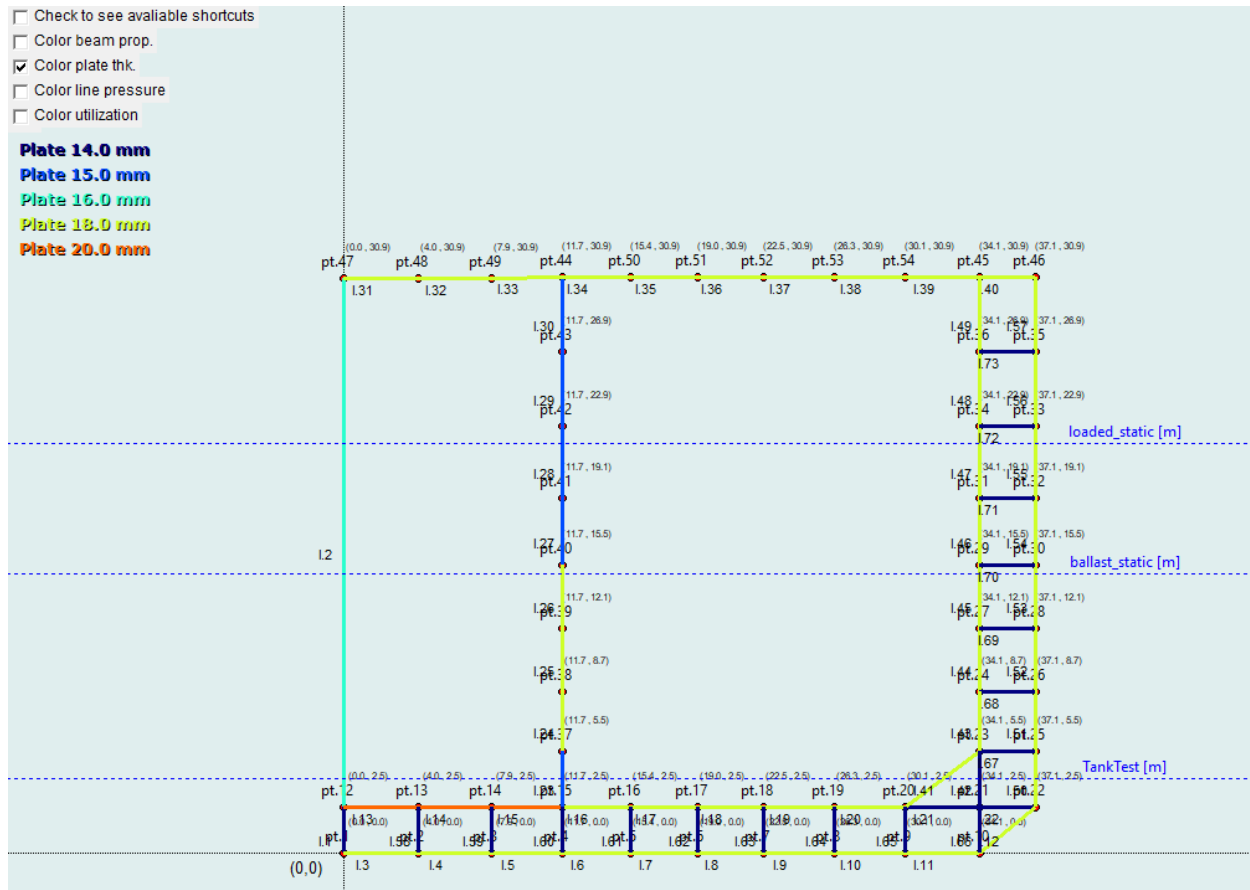
Global stresses (buckling) and structure types

Stresses used in buckling calculations can be reviewed by checking as illustrated next.

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 moduluses: 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 moduluses: 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

A combined utilization can be reviewed using color coding.

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	☑
Check for minimum plate thk.	☑
Check for minimum shear area	☑
Check for buckling (RP-C201)	☑
Check for fatigue (RP-C203)	☑
Check for bow slamming	☐
Check for local stf. buckling	☑
Use weight filter (for speed)	☑

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 "occurence" is a one checked plate/stiffener combination.
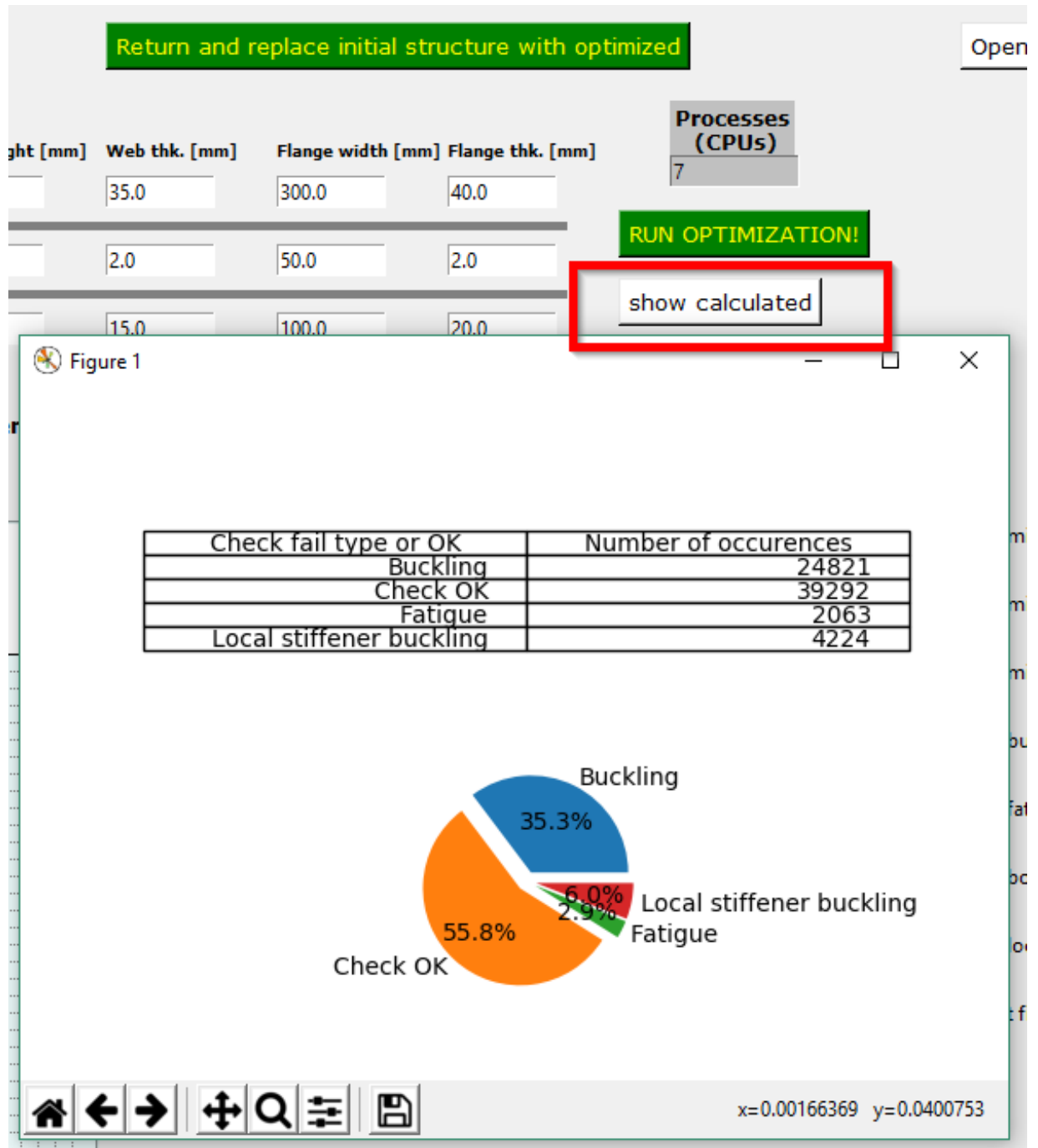
You will also be asked to save to a csv file. If you do not cancel, a csv file will ALL results will pre saved to your chosen location. If you open the file in excel you should see something like show next

Multiple optimization
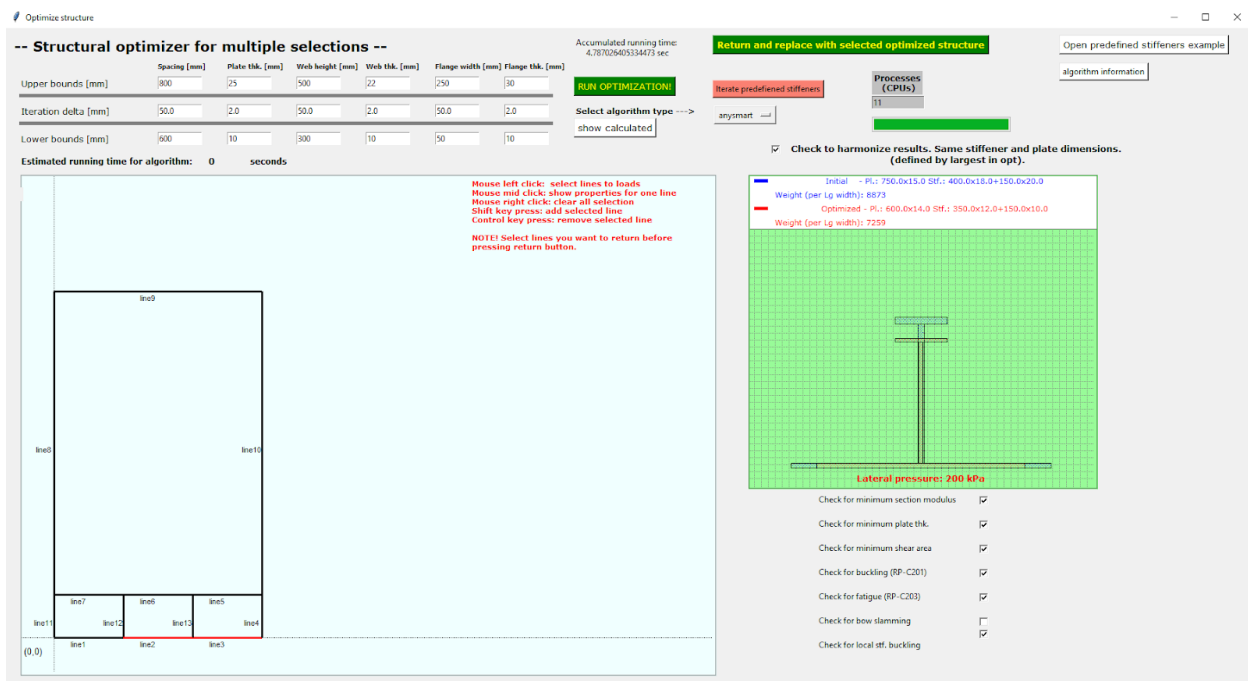
Multiple optimization is done by clicking the "MultiOpt" button.

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. Note that the weight filter is not used when harmonizing, i.e. running will take some more time.

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.
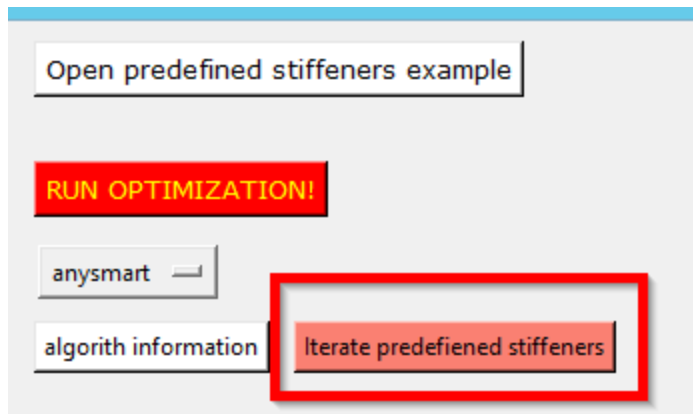
The optimization is started as follows.

1. Start by clicking as illustrated next:
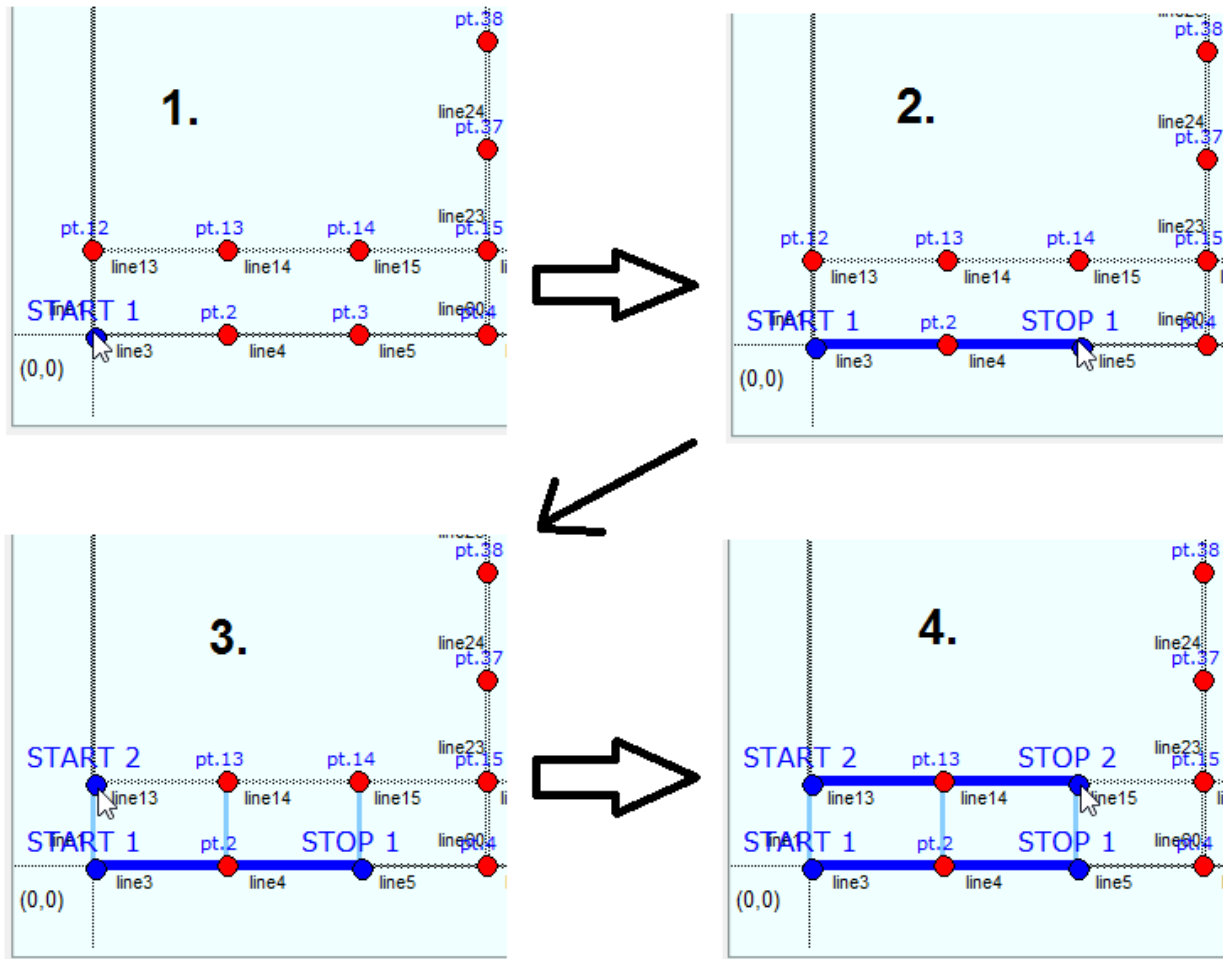
2. Then run optimization.

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
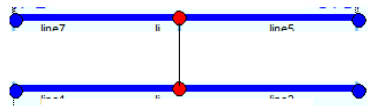
6 plate fields
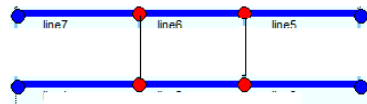
8 plate fields
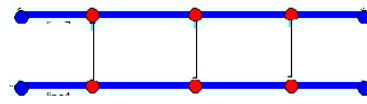
10 plate fields
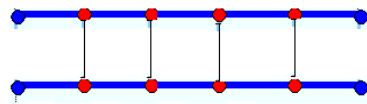
You can, similar to single optimization, select the checks that shall be runned. Also you can set the girder (frame) properties. This is used for calculating the weights.

With reference to the example above, max span mult is the multiplicator 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	☑		**Frame (girder data) for weight calculation:**	
Check for minimum plate thk.	☑ ☑		Girder thickness	0.018
Check for minimum shear area	☑ ☑		Stiffener height	0.25
Check for buckling (RP-C201)	☑		Stiffener thickness	0.015
Check for fatigue (RP-C203)	☑		Stf. flange width	0
Check for bow slamming	☑		Stf. flange thickenss	0
Check for local stf. buckling	☑		For weight calculation of girder: Max span mult / Min span mult	

For weight calculation of girder: Max span mult / Min span mult

1.2	0.8

Maximum span / Minimum span -> | 6 | 2 |

Results are presented as seen next.

RUN OPTIMIZATION!

anysmart

algorith 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.

When the analysis has been runned you should save your results. Just specify a file name in the save file dialog. You can also get detailed individual results for a specified panel. Select number of plate fields in the iteration you want to look at, then choose which panel to get data from. Order of the panels is the same as printed in the left result canvas.

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

Detailed results, printed after running, looks like this :

```
--------------------------------------------------------------------
Plate fields: 22 Frames: 10
pl_650.0x14.0 stf_L180.0x8.0+33.0x17.8 span: 1.0636 structure type: BOTTOM stf. type: L pressure side: p | sigma_y1: 128.6 sigma_y2: 128.6 sigma_x: 177.9 tauxy: 6.4 START 1 OK!
pl_650.0x14.0 stf_L180.0x8.0+33.0x17.8 span: 1.0636 structure type: BOTTOM stf. type: L pressure side: p | sigma_y1: 128.6 sigma_y2: 128.6 sigma_x: 177.9 tauxy: 6.4 ------- OK!
pl_650.0x14.0 stf_L180.0x8.0+33.0x17.8 span: 1.0636 structure type: BOTTOM stf. type: L pressure side: p | sigma_y1: 128.6 sigma_y2: 128.6 sigma_x: 177.9 tauxy: 6.4 ------- OK!
pl_650.0x14.0 stf_L180.0x8.0+33.0x17.8 span: 1.0636 structure type: BOTTOM stf. type: L pressure side: p | sigma_y1: 128.6 sigma_y2: 128.6 sigma_x: 177.9 tauxy: 6.4 ------- OK!
pl_650.0x14.0 stf_L180.0x8.0+33.0x17.8 span: 1.0636 structure type: BOTTOM stf. type: L pressure side: p | sigma_y1: 128.6 sigma_y2: 128.6 sigma_x: 177.9 tauxy: 6.4 ------- OK!
pl_650.0x14.0 stf_L180.0x8.0+33.0x17.8 span: 1.0636 structure type: BOTTOM stf. type: L pressure side: p | sigma_y1: 128.6 sigma_y2: 128.6 sigma_x: 177.9 tauxy: 6.4 ------- OK!
pl_650.0x14.0 stf_L180.0x8.0+33.0x17.8 span: 1.0636 structure type: BOTTOM stf. type: L pressure side: p | sigma_y1: 128.6 sigma_y2: 128.6 sigma_x: 177.9 tauxy: 6.4 ------- OK!
pl_650.0x14.0 stf_L180.0x8.0+33.0x17.8 span: 1.0636 structure type: BOTTOM stf. type: L pressure side: p | sigma_y1: 128.6 sigma_y2: 128.6 sigma_x: 177.9 tauxy: 6.4 ------- OK!
pl_650.0x14.0 stf_L160.0x8.0+36.0x18.7 span: 1.0636 structure type: BOTTOM stf. type: L pressure side: p | sigma_y1: 128.6 sigma_y2: 128.6 sigma_x: 174.9 tauxy: 6.4 ------- OK!
pl_650.0x14.0 stf_L160.0x8.0+36.0x18.7 span: 1.0636 structure type: BOTTOM stf. type: L pressure side: p | sigma_y1: 128.6 sigma_y2: 128.6 sigma_x: 170.0 tauxy: 6.4 -END 1- OK!
pl_750.0x14.0 stf_L160.0x8.0+36.0x18.7 span: 1.0636 structure type: GENERAL_INTERNAL_WT stf. type: L pressure side: p | sigma_y1: 115.7 sigma_y2: 115.7 sigma_x: 144.9 tauxy: 6.4 START 2 OK!
pl_750.0x14.0 stf_L160.0x8.0+36.0x18.7 span: 1.0636 structure type: GENERAL_INTERNAL_WT stf. type: L pressure side: p | sigma_y1: 115.7 sigma_y2: 115.7 sigma_x: 144.9 tauxy: 6.4 ------- OK!
pl_750.0x14.0 stf_L160.0x8.0+36.0x18.7 span: 1.0636 structure type: GENERAL_INTERNAL_WT stf. type: L pressure side: p | sigma_y1: 115.7 sigma_y2: 115.7 sigma_x: 144.9 tauxy: 6.4 ------- OK!
pl_750.0x14.0 stf_L160.0x8.0+36.0x18.7 span: 1.0636 structure type: GENERAL_INTERNAL_WT stf. type: L pressure side: p | sigma_y1: 115.7 sigma_y2: 115.7 sigma_x: 144.9 tauxy: 6.4 ------- OK!
pl_750.0x14.0 stf_L160.0x8.0+36.0x18.7 span: 1.0636 structure type: GENERAL_INTERNAL_WT stf. type: L pressure side: p | sigma_y1: 115.7 sigma_y2: 115.7 sigma_x: 144.9 tauxy: 6.4 ------- OK!
pl_750.0x14.0 stf_L160.0x8.0+36.0x18.7 span: 1.0636 structure type: GENERAL_INTERNAL_WT stf. type: L pressure side: p | sigma_y1: 115.7 sigma_y2: 115.7 sigma_x: 144.9 tauxy: 6.4 ------- OK!
pl_750.0x14.0 stf_L160.0x8.0+36.0x18.7 span: 1.0636 structure type: GENERAL_INTERNAL_WT stf. type: L pressure side: p | sigma_y1: 115.7 sigma_y2: 115.7 sigma_x: 144.9 tauxy: 6.4 ------- OK!
pl_750.0x14.0 stf_L160.0x8.0+36.0x18.7 span: 1.0636 structure type: GENERAL_INTERNAL_WT stf. type: L pressure side: p | sigma_y1: 115.7 sigma_y2: 115.7 sigma_x: 144.9 tauxy: 6.4 ------- OK!
pl_750.0x14.0 stf_L160.0x8.0+36.0x18.7 span: 1.0636 structure type: GENERAL_INTERNAL_WT stf. type: L pressure side: p | sigma_y1: 115.7 sigma_y2: 115.7 sigma_x: 144.9 tauxy: 6.4 ------- OK!
pl_750.0x14.0 stf_L160.0x8.0+36.0x18.7 span: 1.0636 structure type: GENERAL_INTERNAL_WT stf. type: L pressure side: p | sigma_y1: 115.7 sigma_y2: 115.7 sigma_x: 144.9 tauxy: 6.4 -END 2- OK!
Weight details for this solution:
Weight of main structure: ['1423.0', '1423.0', '1423.0', '1423.0', '1423.0', '1423.0', '1423.0', '1423.0', '1423.0', '1413.8', '1413.8', '1381.1', '1381.1', '1381.1', '1381.1', '1381.1', '1
Weight of frames:          ['4172.8', '4172.8', '4172.8', '4172.8', '4172.8', '4172.8', '4040.3', '4040.3', '4040.3', '4040.3']
Scales used on frames:     ['0.9', '0.9', '0.9', '0.9', '0.9', '0.9', '0.9', '0.9', '0.9', '0.9']
--------------------------------------------------------------------
```
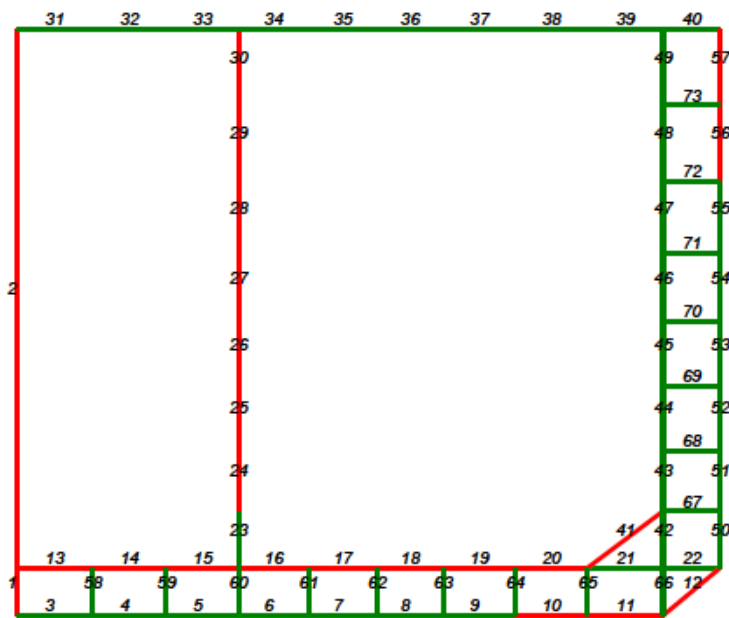
Reporting

A pdf report can be created by clicking "Reporting - Generate PDF report". The report will include all information for all lines. An example is seen next.
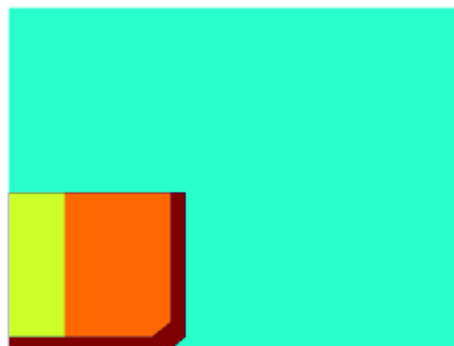

ANYstructure report generator

User: CEFANY

Time : Fri, 16 Apr 2021 17:31:48 +0000

Compartments:

Compartments returned from search operation displayed below

Name: comp2, content: crude_oil
Min. elevation: 2.5, Max. elevation: 30.9
Applied overpressure: 25000.0
(a_stat, a_dyn_loa, a_dyn_bal): (9.81, 3.0, 3.0)

Name: comp3, content: crude_oil
Min. elevation: 2.5, Max. elevation: 30.900000000000002
Applied overpressure: 25000.0
(a_stat, a_dyn_loa, a_dyn_bal): (9.81, 3.0, 3.0)

Name: comp4, content: ballast
Min. elevation: 0.0, Max. elevation: 30.900000000000002
Applied overpressure: 25000.0
(a_stat, a_dyn_loa, a_dyn_bal): (9.81, 3.0, 3.0)

*area calculation inaccuracies due to thickness of barriers (BHD/Deck)

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.
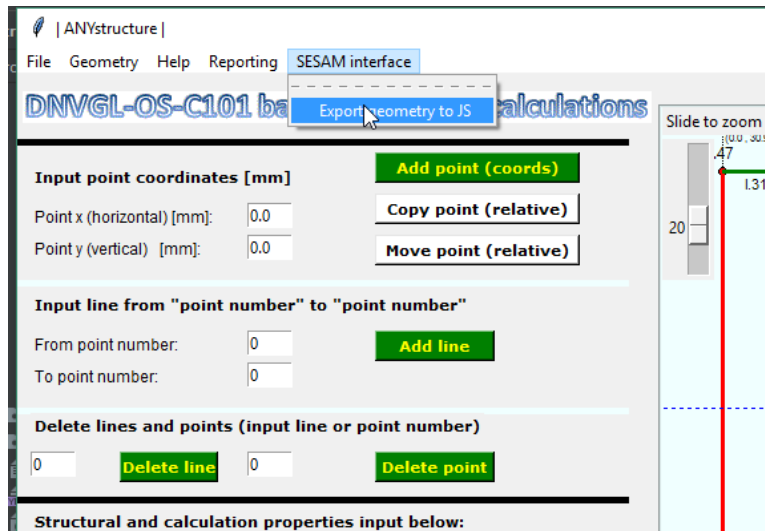
The result is illustrated below: