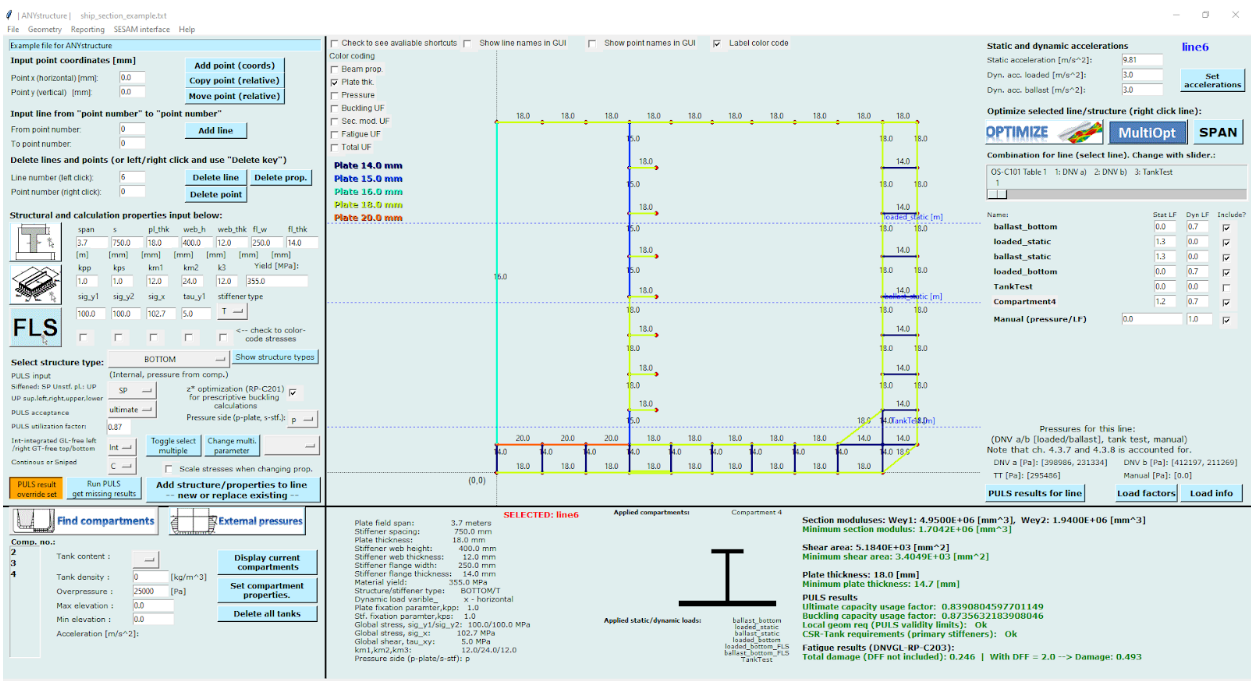
ANYstructure



Documentation

2021

Version 3.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 DNV 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**
- **PULS (Panel Ultimate Limit State)**
- DNVGL-CG-0128 Buckling

Modelling

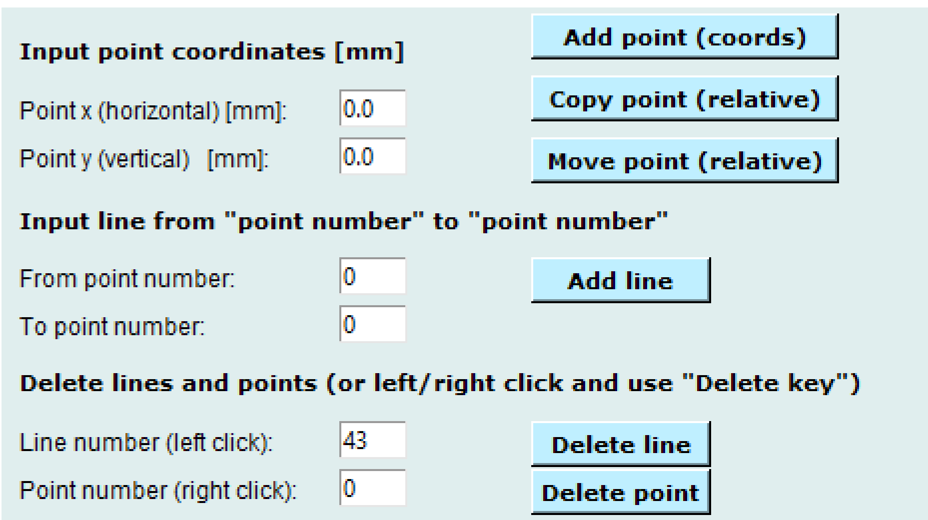
Modelling is done in upper left corner.

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)



Speed up your modelling significantly 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

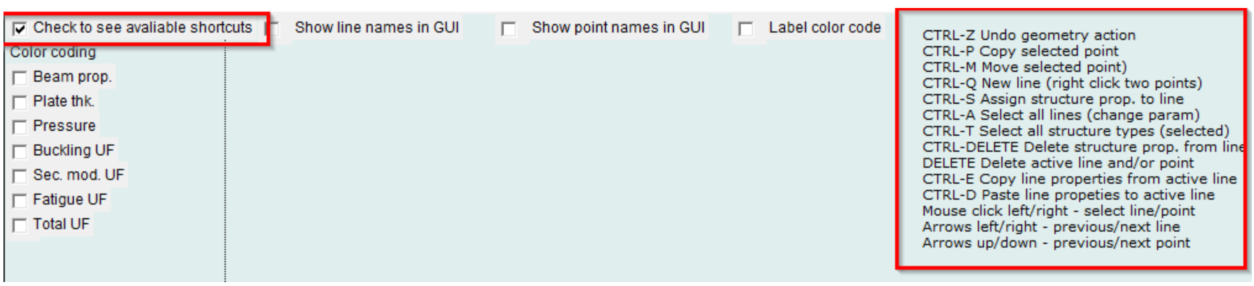
Arrows up/down Toggle point in model

Arrows left/right Toggle lines in model

CTRL-A Select all lines in model for changing a selected parameter for all

CTRL-T Select all lines of a specific structure type for changing a parameter for multiple lines.

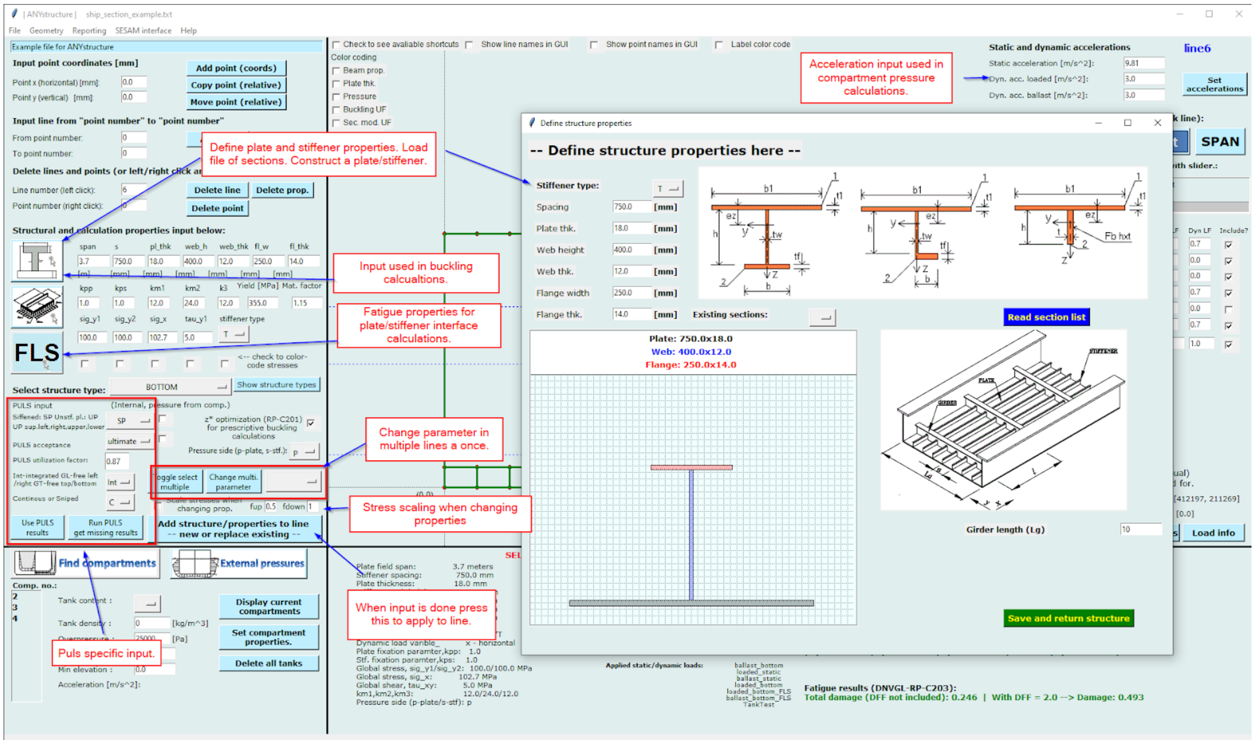
The shortcuts can be shown in the GUI as seen next.



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”.

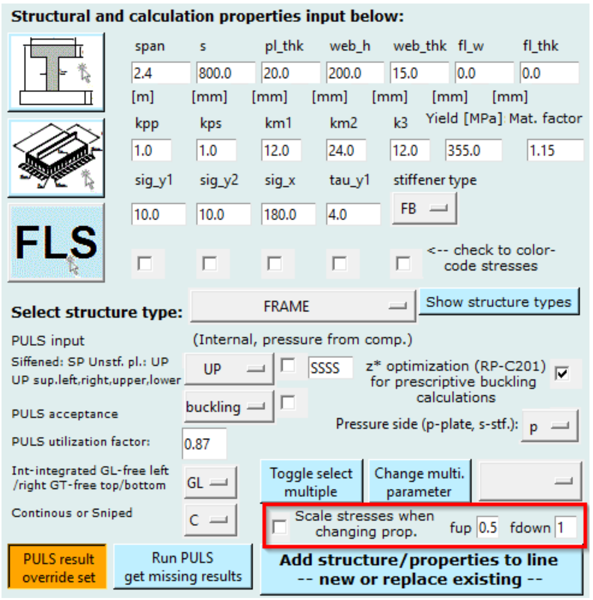


By default z* is ticked. This affects the buckling results and will generally give lower utilization than using z* = 0. See description below.

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.

Scale stresses

Stresses can be automatically scaled when changing a property, for example plate thickness. The parameters fup and fdown specify the factor to be applied to the scaling when scaling up (thicker plate) or down (thinner plate).



The formula applied is referenced next. The factor depends on your case.

If panel thickness (T) is changed (dT), stress may be scaled by a factor (f) according to the formula:

$$newStress = \left(\frac{T}{T + f * dT}\right) * oldStress$$

- f = 0.0 -> stress does not depend on local thickness change.
- f = 1.0 -> stress is proportional to local thickness change.

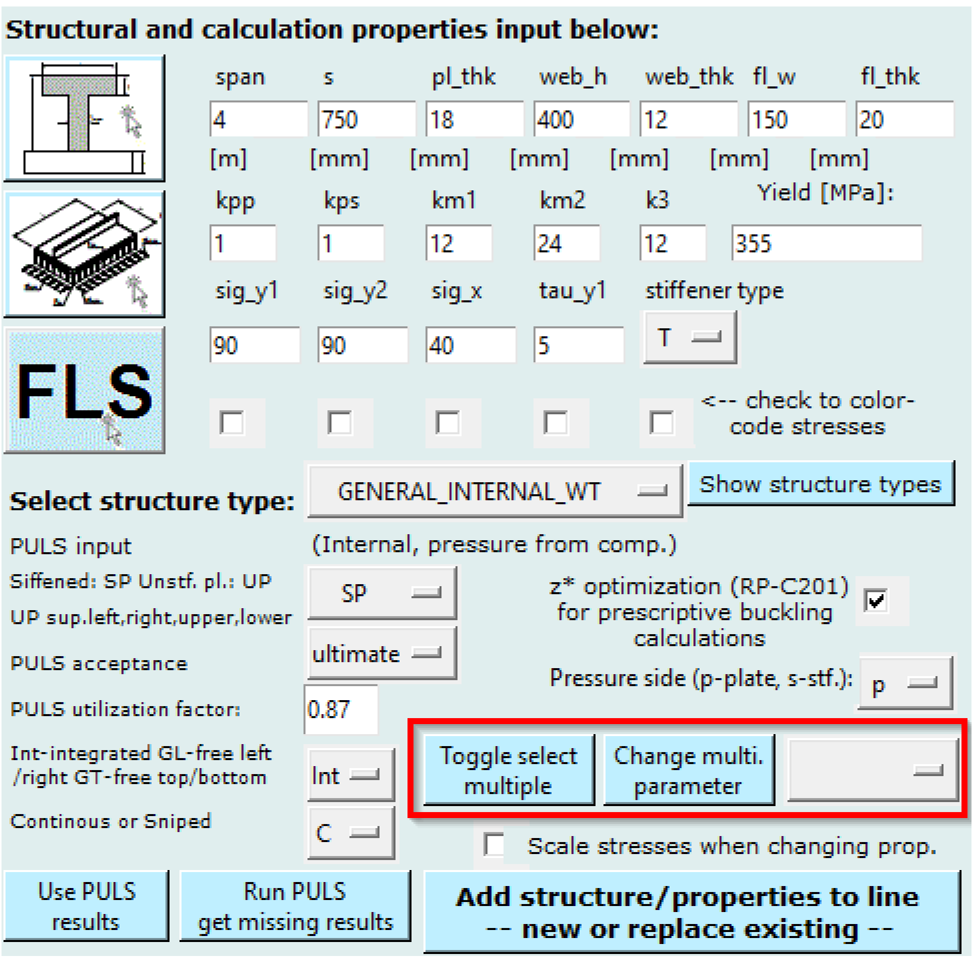
Stresses to be scaled are axial stress (SigmaX), transversal stress(SigmaY) and shear stress (TauXY).

The parameters fdown = 1 and fup = 0.5 is by default. The general idea is that it is conservative to accept lower stress reduction when increasing thicknesses.

Changing multiple properties

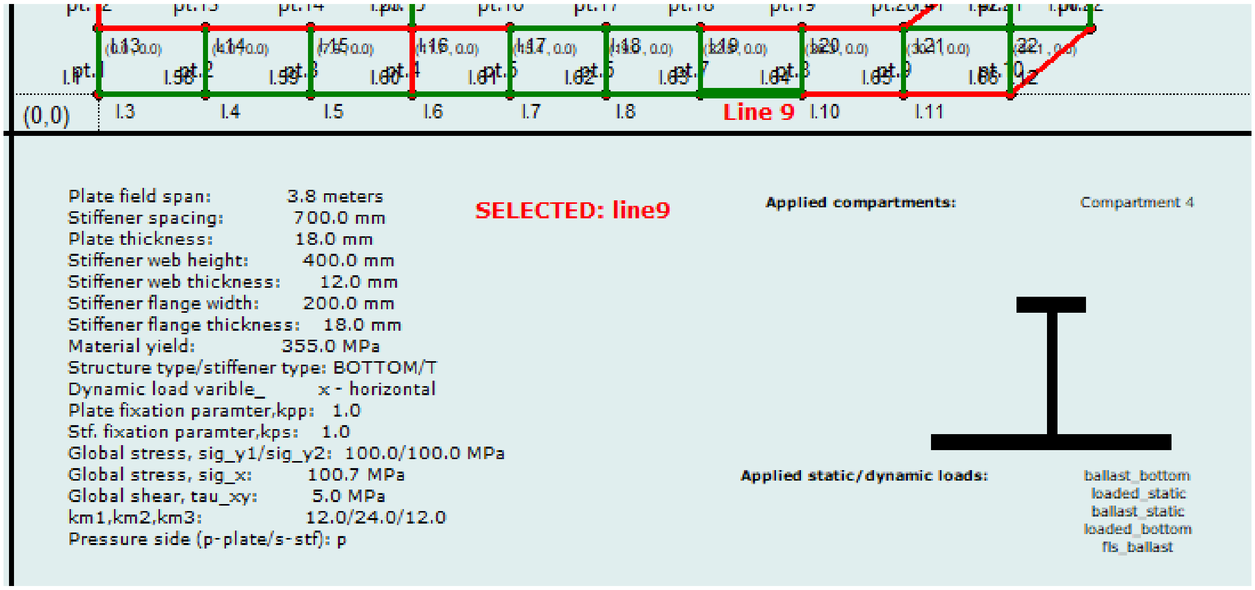
If you want to change a single property for multiple lines. How to do it:

1. Press Toggle select multiple
2. Select the parameter to change
3. Select the lines to change. Click single lines, CTRL-A or CTRL-T (see shortcuts)
4. Press Change multi. param.



Display properties

If you click a line properties is displayed in the window below as seen next.



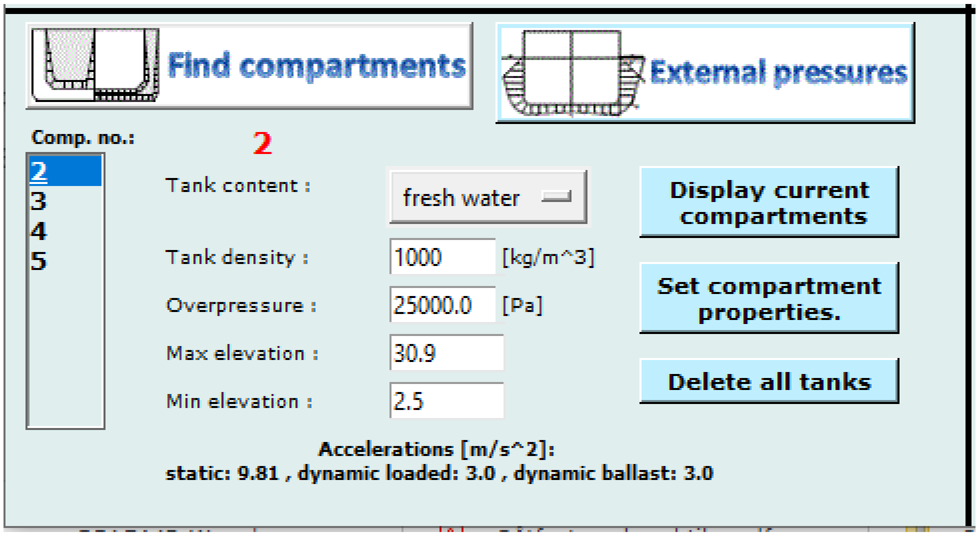
Also it is recommended to use color coding to review your input.

Define tanks

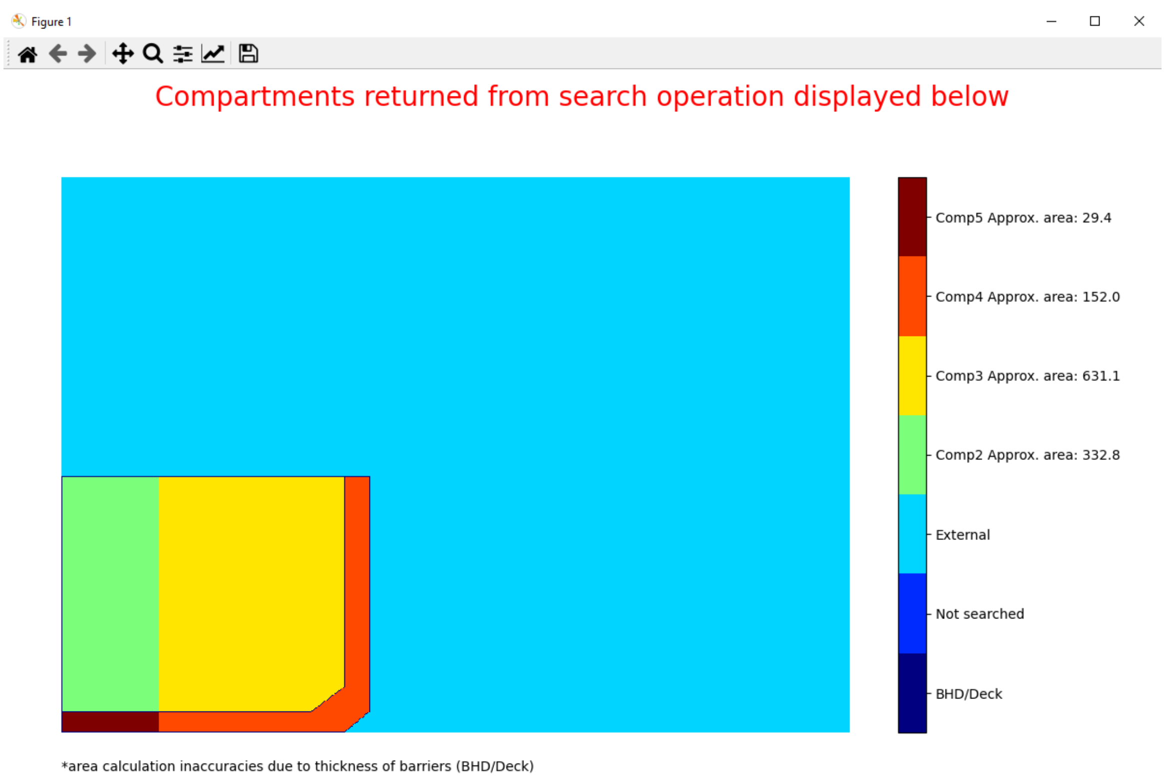
Tanks are searched for when clicking “Find compartments”. Non watertight structure are ignored. For information on structure types click “Show structure types”.

<u>By default tank content density is set to 0.</u>

Ather tanks are found content and overpressure must be defined as seen next.

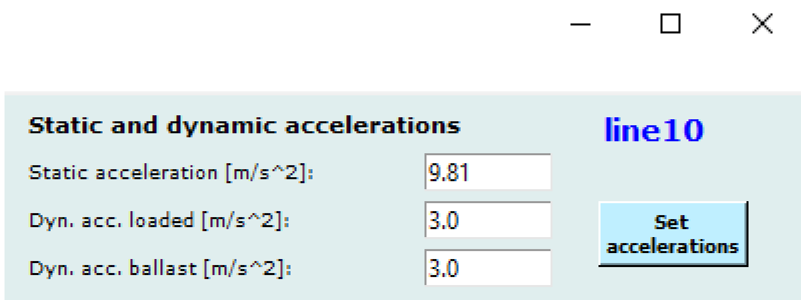


If you press "Display current compartments" after doing a compartment search, the result of the search is illustrated as seen next. Approximate area of the respective compartments is also shown.



Setting accelerations

Accelerations applies to tank content. I is set in the upper right corner as seen next.



Define external pressures

Click "External pressures" to define pressures acting on the structures.

NOTE:

FOR DYNAMIC EQUATION THE FOLLOWING APPLIES

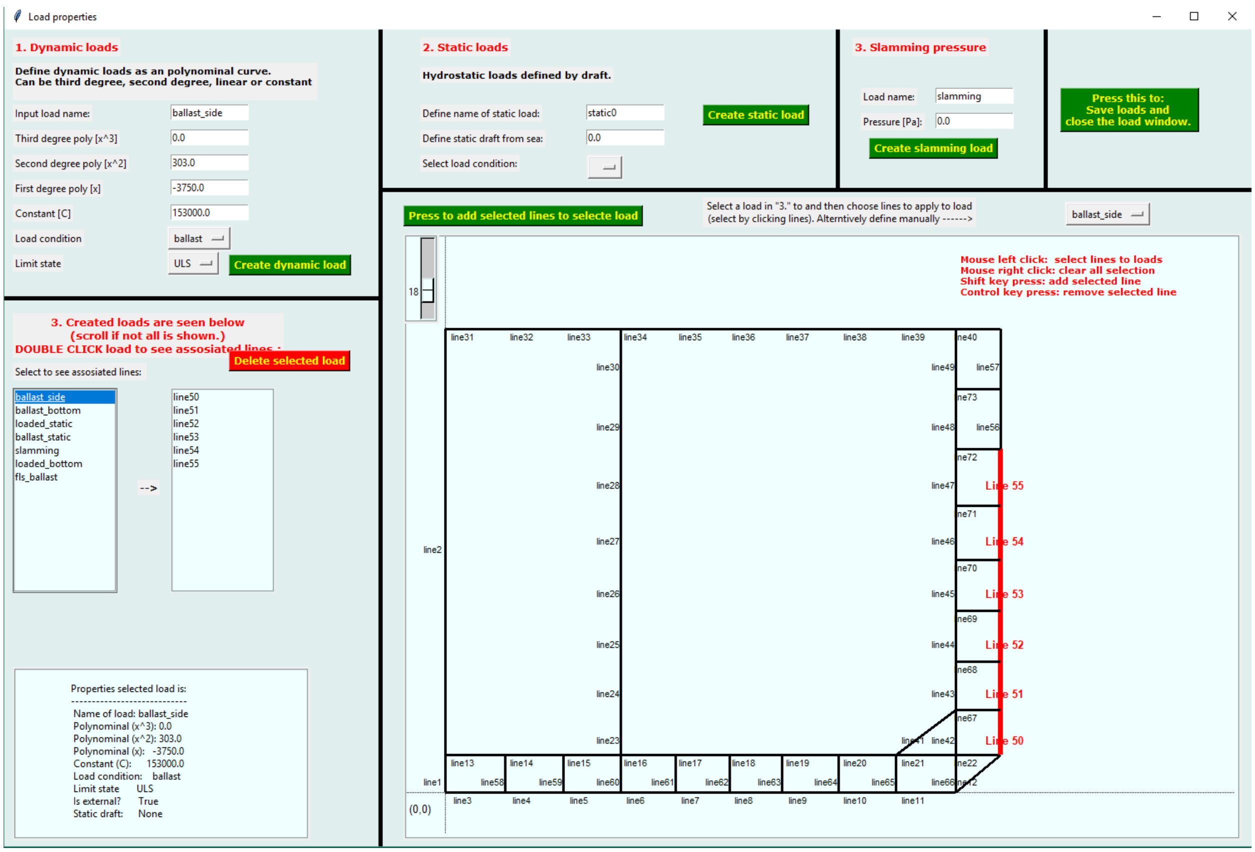
X (horizontal) used for BOTTOM, BBT, HOPPER, MD

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.**

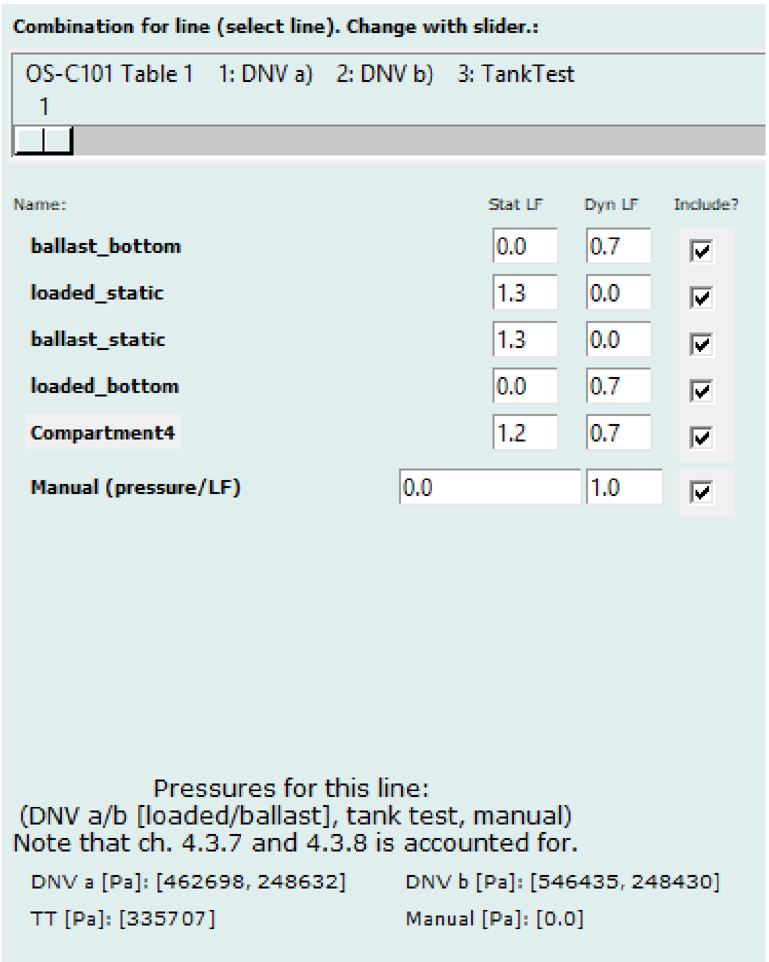


Load combinations

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.**

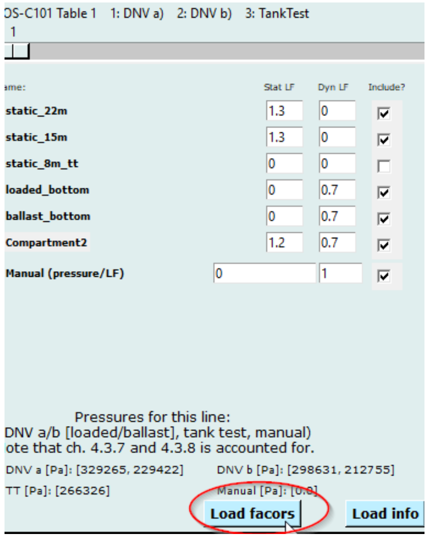
3. **You can deselect a load by manually inputting load factor to 0 or deselect include.**

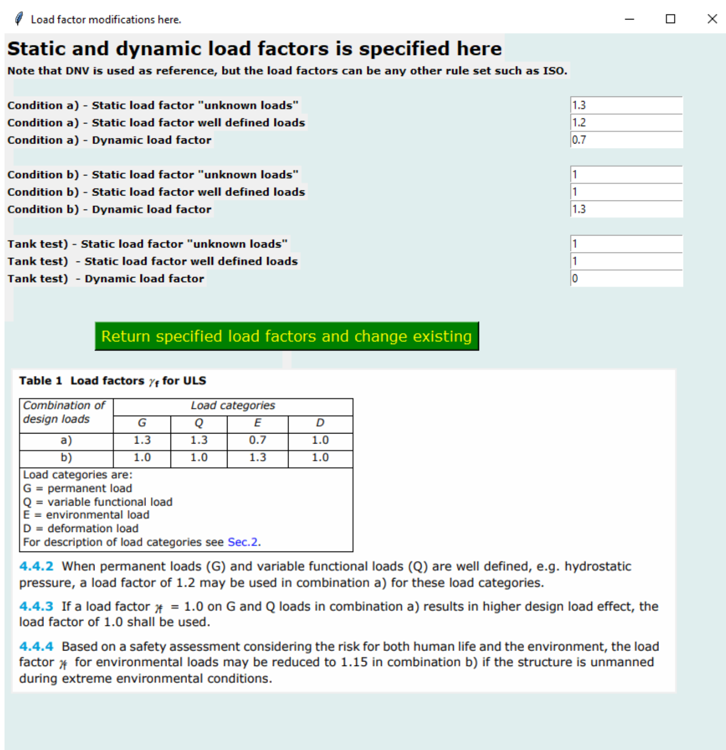


Changing load factors

You can change default load factors and existing load factors using the button seen in the next illustration.

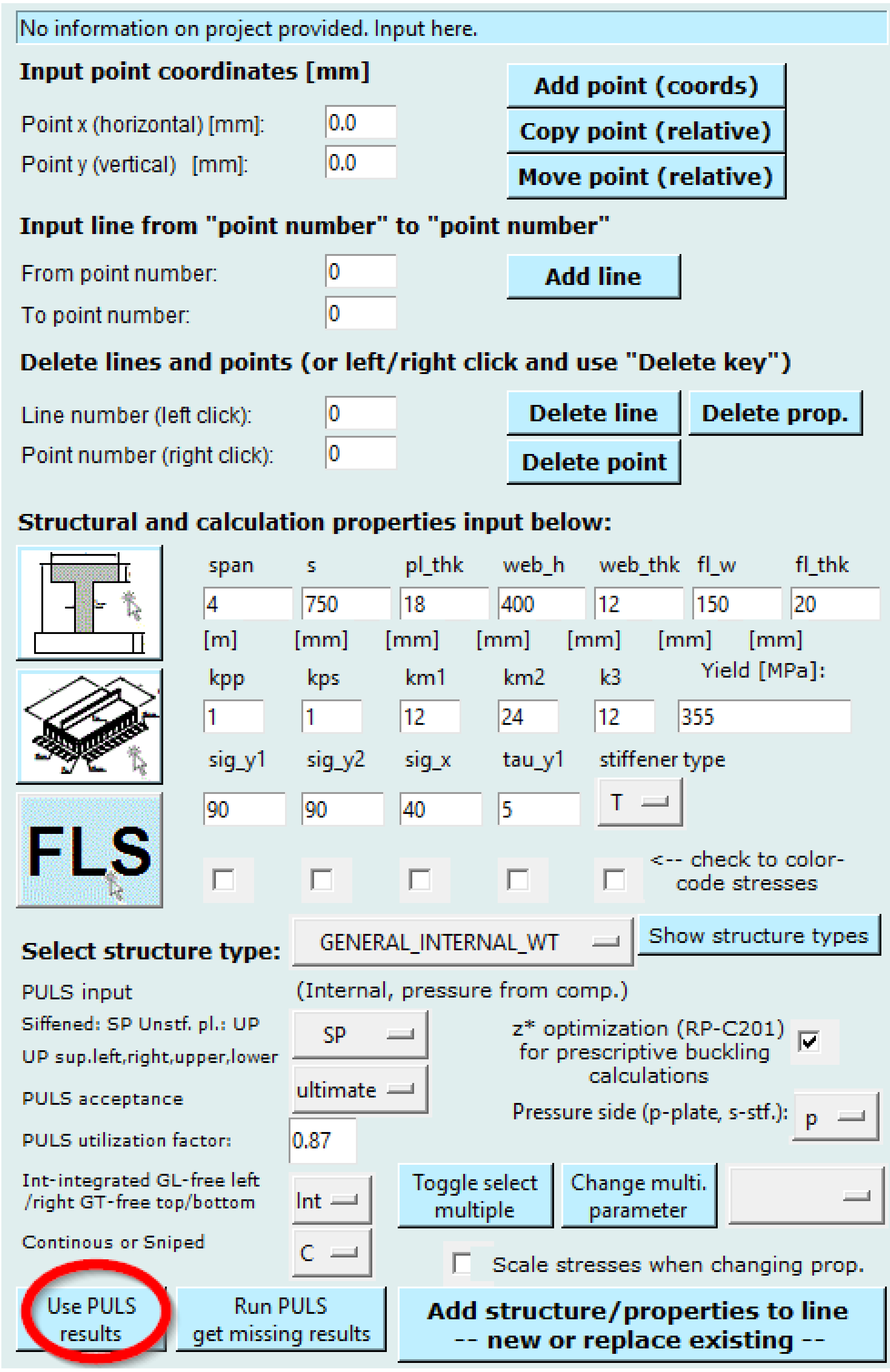
Load factors are based on standard DNV LRFD factors, but any values can be used.





PULS integration

ANYstructure can use PULS software to calculate buckling. PULS is a licensed DNV software. Consequently, PULS integration will not work if you do not have the license. Specifically ANYstructure uses the PULS Excel sheet to calculate. Macros must be enabled for the sheet. The sheet may require a 32 bit version of Microsoft Office. Using PULS is activated by clicking the button seen next. When running a line for the first time, you will be asked to provide the location of the PULS excel sheet. The sheet should be empty and macros should be enabled.



PULS parameters are set for each line.

1. Stiffened panel (SP) or unstiffened plate (UP).
    a. If UP is chosen you can specify the boundary conditions. The conditions consist of four letters, representing left side, right side, top and bottom (in this order). ‘S’ means simply supported and ‘C’ means Clamped. ‘SSSS’ is conseqeuntly all simply supported and for example ‘SSCC’ is simply supported sides with clamped top and bottom.
2. Integrated (Int) or girder panels (GL/GT)
3. Continuous or Sniped stiffener
4. Ultimate or buckling acceptance. In general ultimate acceptance is more representative for larger plate fields where loads can be redistributed. Reference is made to DNV standards.

For theory check out PULS manual and/or Part 1, chapter 8 of the IACS Common structural rules for bulk carriers and oil tankers:

https://iacs.org.uk/publications/common-structural-rules/csr-for-bulk-carriers-and-oil-tankers/

Detailed PULS results can be viewed by selecting a line a pressing the “PULS results for line” button:

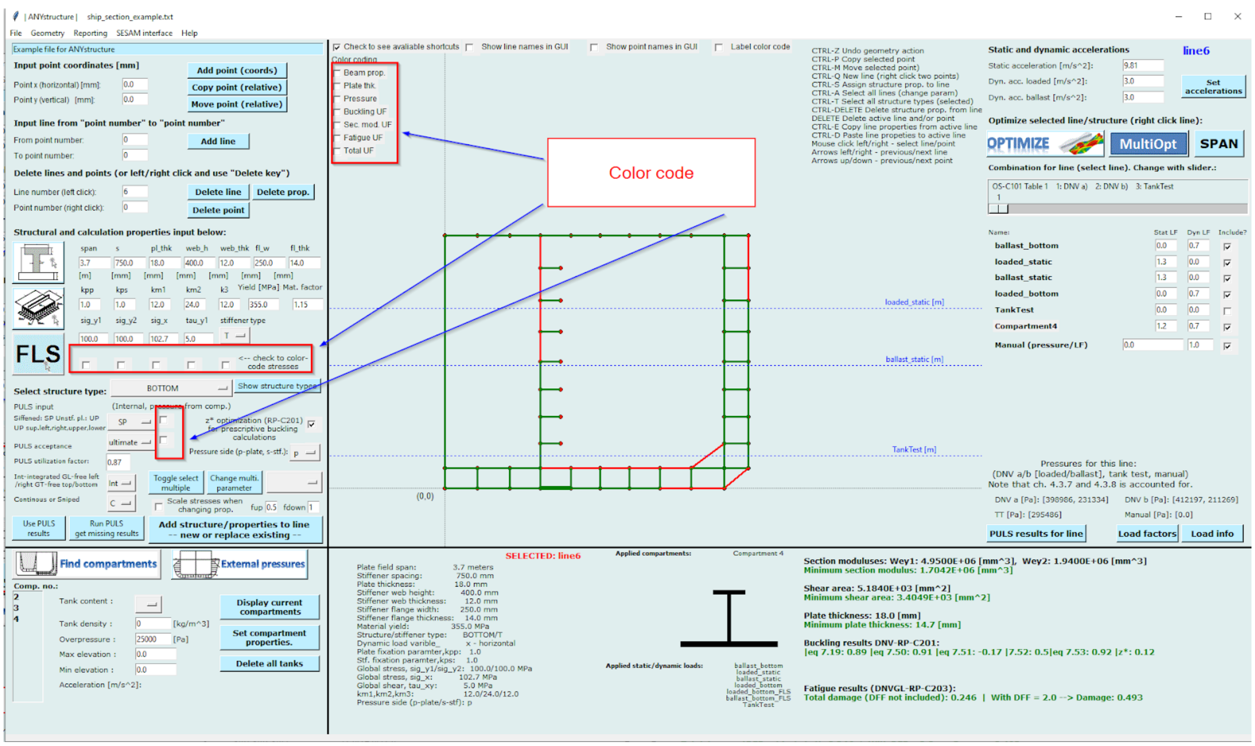
| ANYstructure | ship_section_example.txt

```
Identification : line8
Plate geometry
   Length of panel : 3500.0 mm
   Stiffener spacing : 750.0 mm
   Plate thick. : 18.0 mm
Primary stiffeners
   Number of stiffeners : 10.0
   Stiffener type : T-bar
   Stiffener boundary : Cont
   Stiff. Height : 400.0 mm
   Web thick. : 12.0 mm
   Flange width : 250.0 mm
   Flange thick. : 12.0 mm
   Flange ecc. : 0.0 mm
   Tilt angle : 0.0 degrees
Secondary stiffeners
   Number of sec. stiffeners : 0.0
   Secondary stiffener type : Flatbar
   Stiffener boundary : SS
   Stiff. Height : 0.0 mm
   Web thick. : 0.0 mm
   Flange width : 0.0 mm
   Flange thick. : 0.0 mm
Model imperfections
   Imp. level : Default
   Plate : 3.75 mm
   Stiffener : 3.5 mm
   Stiffener tilt : 3.5 mm
Material
   Modulus of elasticity : 210000.0 MPa
   Poisson's ratio : 0.3
   Yield stress plate : 355.0 MPa
   Yield stress stiffener : 355.0 MPa
Aluminium prop
   HAZ pattern : -
   HAZ red. factor : -
Applied loads
   Axial stress : 102.0 MPa
   Trans. stress : 100.0 MPa
   Trans. stress 2 : 100.0 MPa
   Shear stress : 5.0 MPa
   Pressure (fixed) : 0.438508 MPa
Bound cond.
   In-plane support : Integrated
Global elastic buckling
   Axial stress : 367.0 MPa
   Trans. Stress : 362.0 MPa
   Trans. stress : 362.0 MPa
   Shear stress : 18.0 MPa
Local elastic buckling
   Axial stress : 134.0 MPa
   Trans. Stress : 132.0 MPa
   Trans. stress : 132.0 MPa
   Shear stress : 7.0 MPa
Ultimate capacity
   Actual usage Factor : 0.73
   Allowable usage factor : 1.0
   Status : Ok
Failure modes
   Plate buckling : 37.0 %
```

Reviewing data

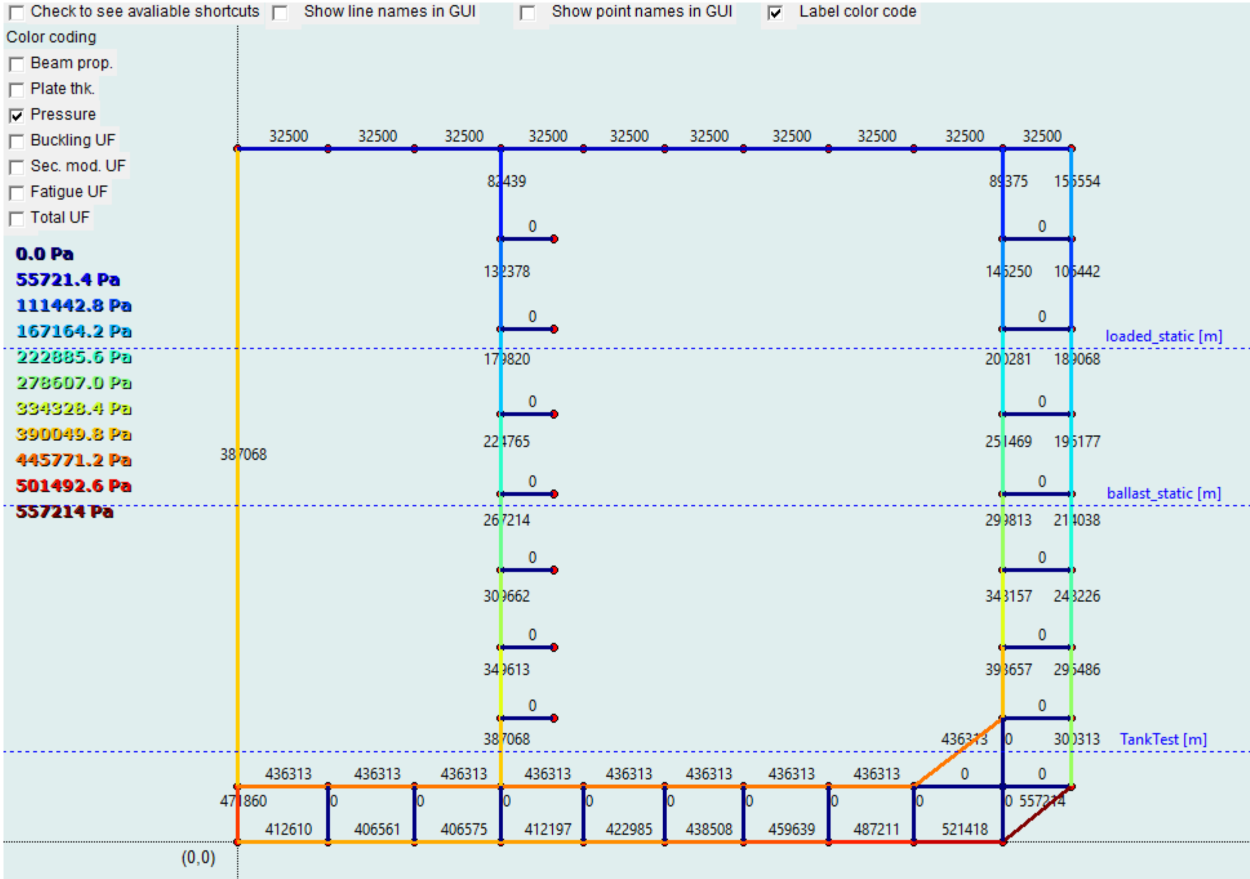
Color coding

All color coding options are indicated next.



Loads

Pressure magnitude can be reviewed by using color coding. The highest total pressure used in calculations is shown.



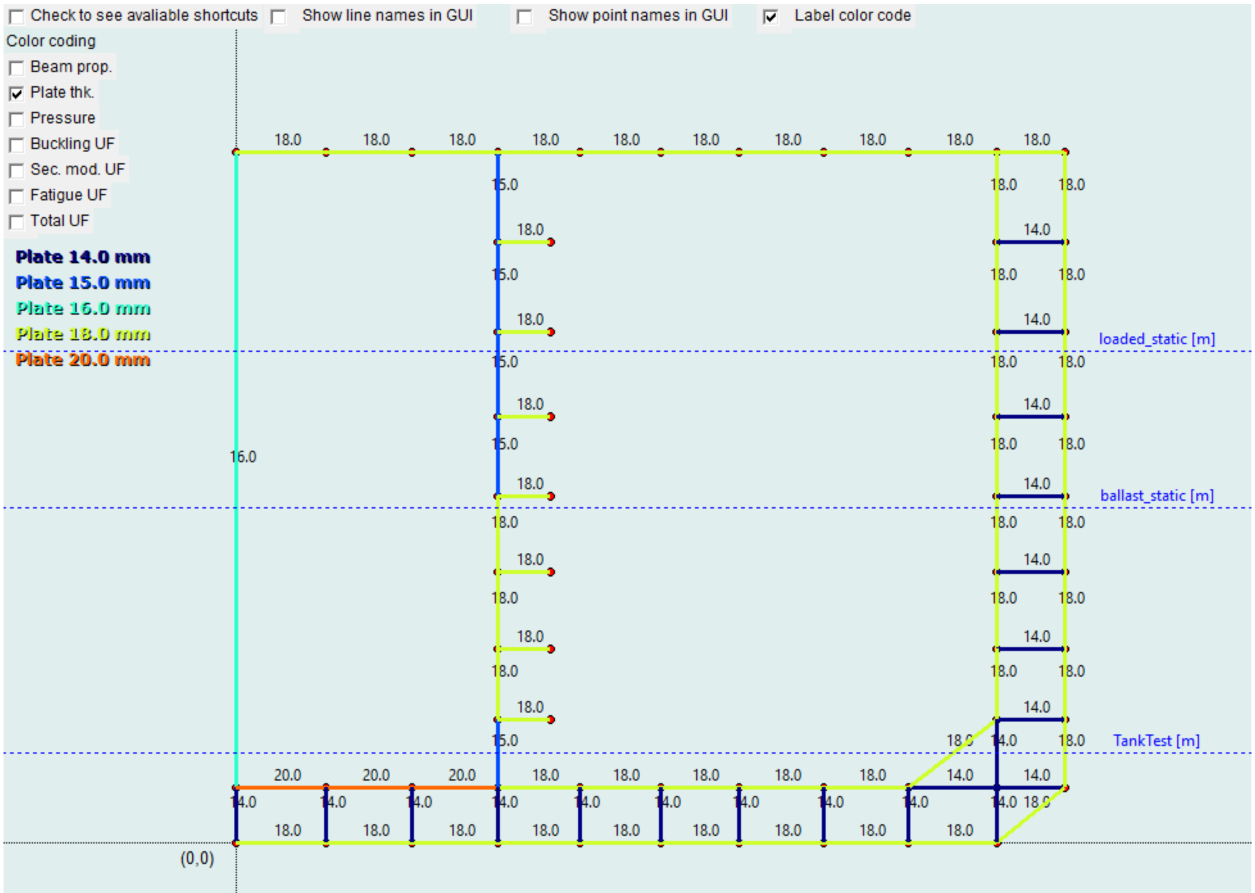
Load calculations and results can be reviewed by clicking the “Load info” button. An example is seen in the next illustration.

| ANYstructure | ship_section_example.txt

```
Load results for line8

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*181077.2 = 126754.1

RESULT: 287580.2 + 126754 = 414334.2
------------------------------------------------------------------
Loads for condition: ballast - dnva
 dynamic with acceleration: 3.0 is:
 1*0.7*57425.2 = 40197.6
 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: 40197.6 + 196077 = 236275.0
------------------------------------------------------------------
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*1.2*181077.2 = 217292.7

RESULT: 221215.5 + 217293 = 438508.2
------------------------------------------------------------------
Loads for condition: ballast - dnvb
 dynamic with acceleration: 3.0 is:
 1*1.2*57425.2 = 68910.2
 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: 68910.2 + 150829 = 219739.0
------------------------------------------------------------------
Tank test for: t
1 * 1.0 * 40221.0 + 0 = 40221
Tank test for: comp4
1 * 1.0 * 310707.2 + 25000.0 * 1 = 335707
Manual pressure:
0.0 * 1.0 * 1 = 0.0
```

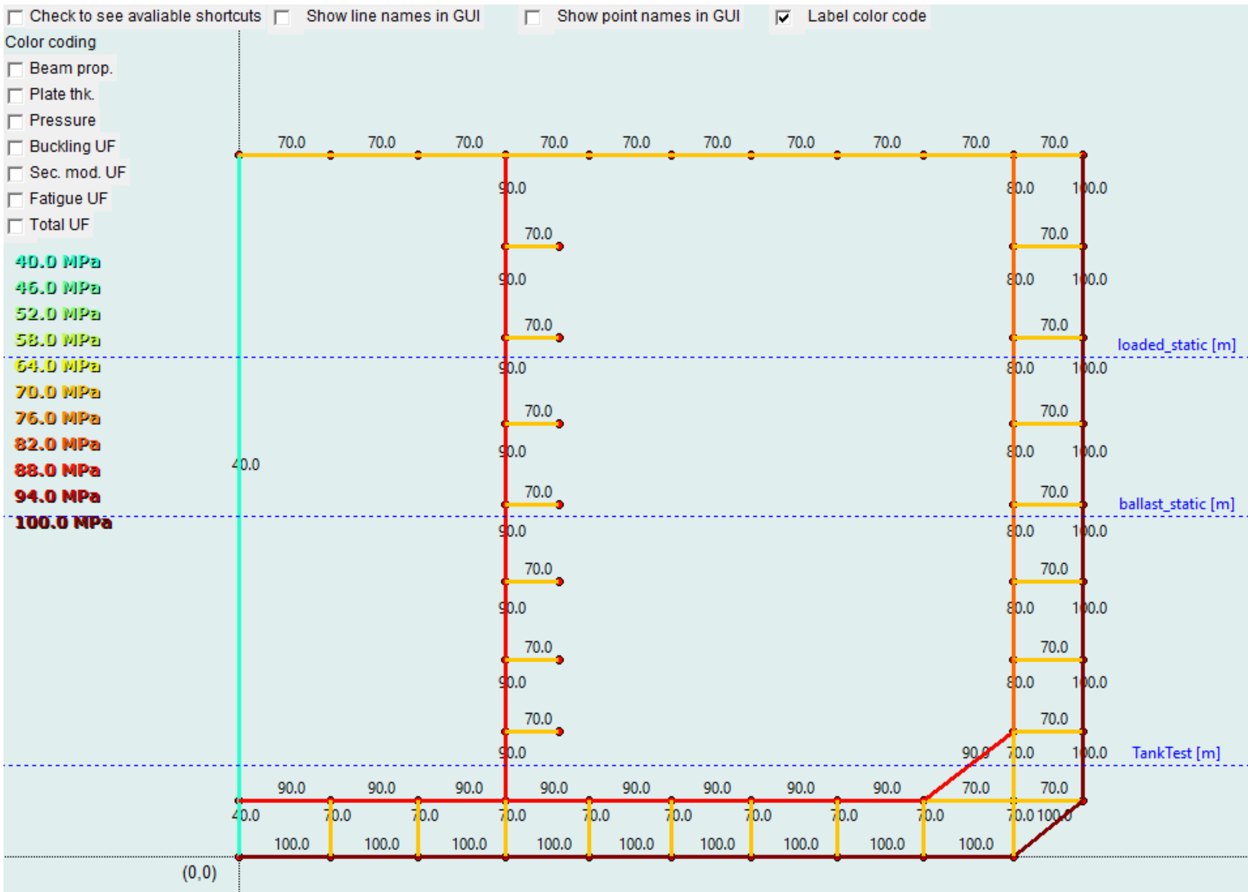
Thickness and beam properties

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

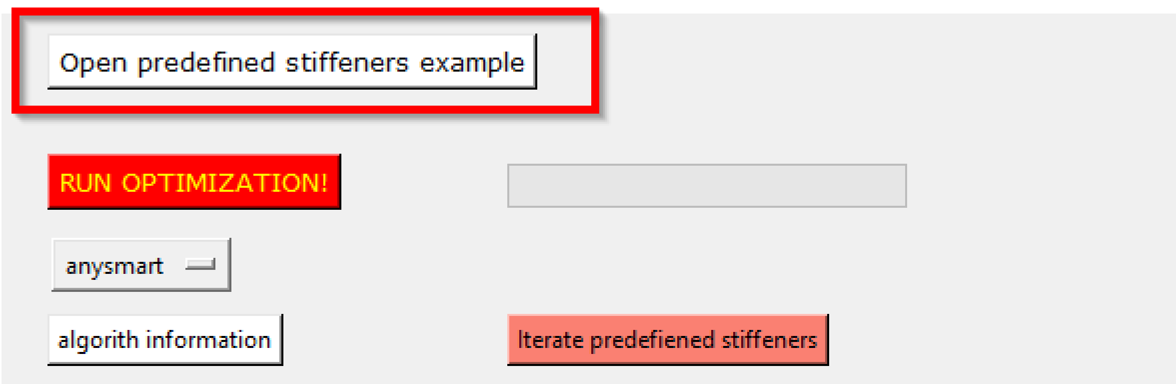
A combined utilization can be reviewed using color coding.

Optimization

Optimization iteration by predefined stiffeners

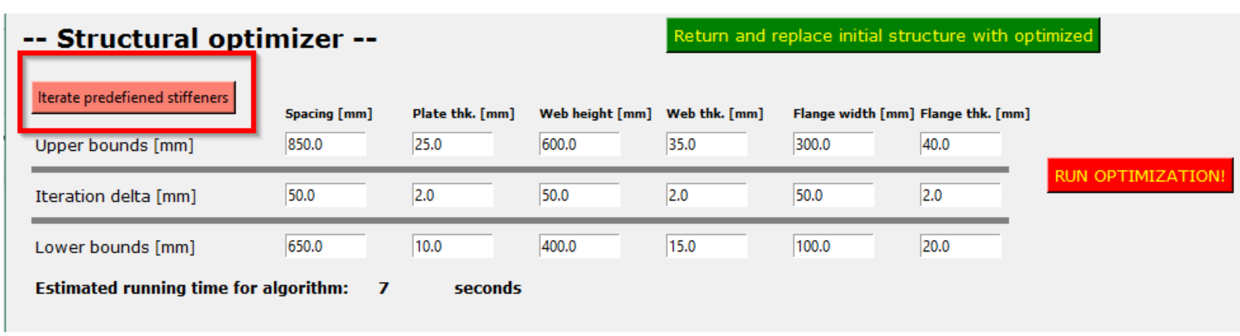
From 0.5 you can iterate by a defined set of stiffeners. Press the button marked below. Open a csv (or json) file. Then start your iterations. The only other input is the stiffener spacing and plate thickness.

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

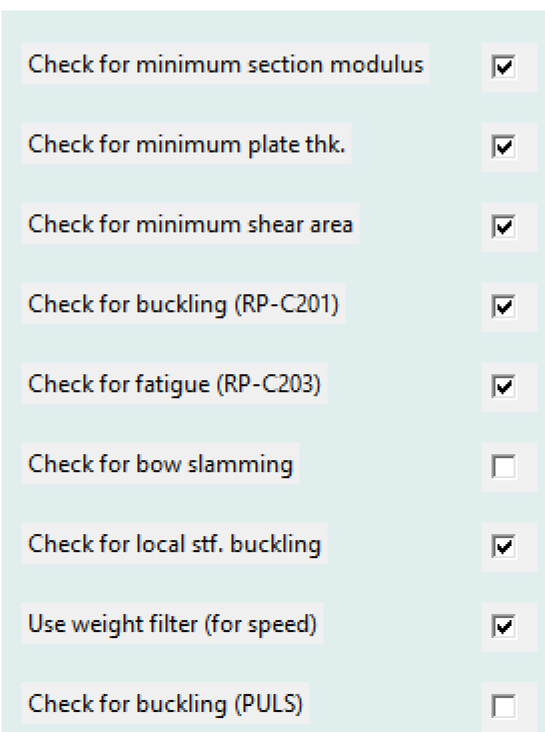
Single optimization is done by clicking a line and clicking the “OPTIMIZE” button.

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**

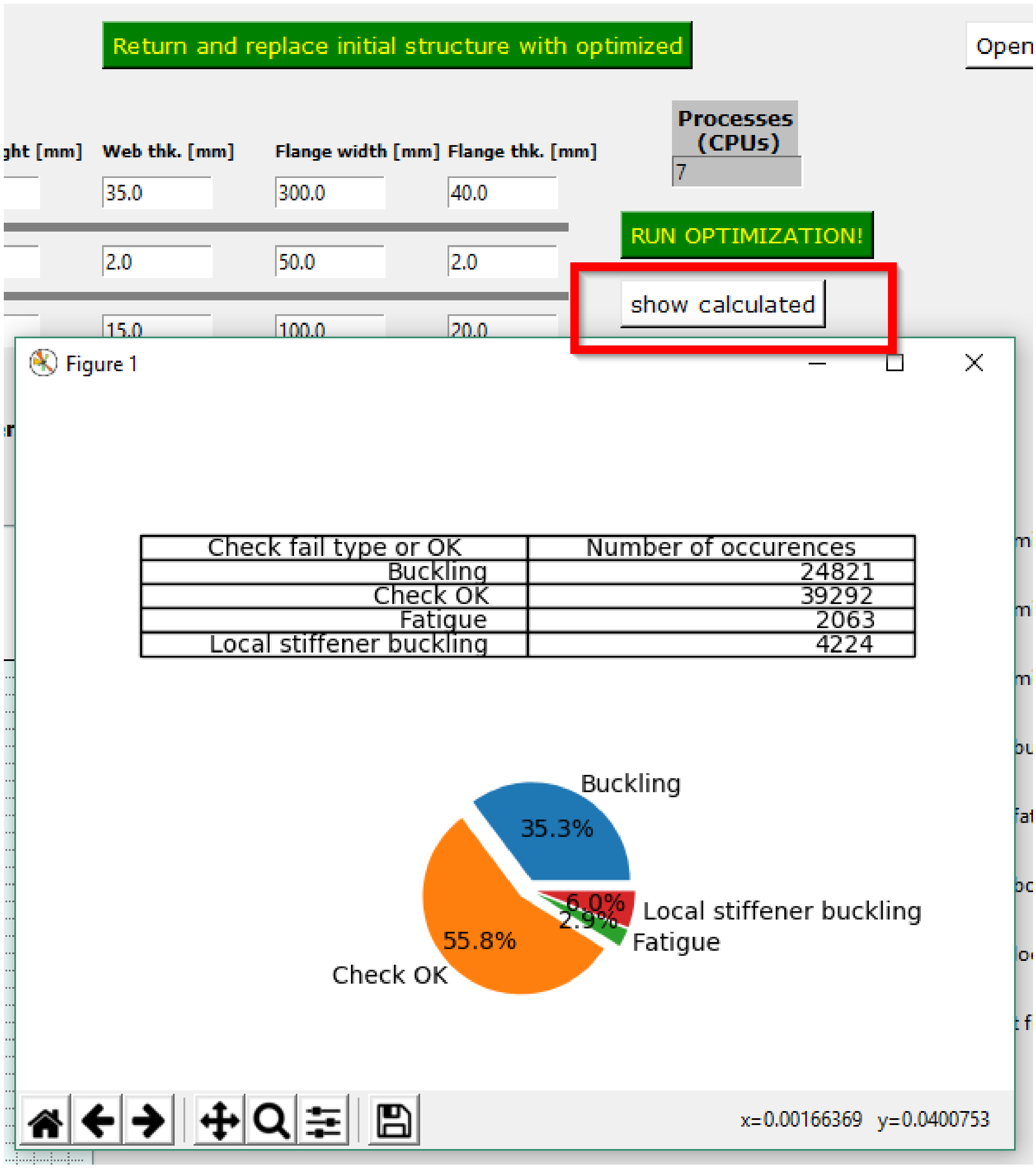
Various checks in the optimization module:

You can select the checks to be performed. PULS buckling can be used in optimization. Remember to check the running time.

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.

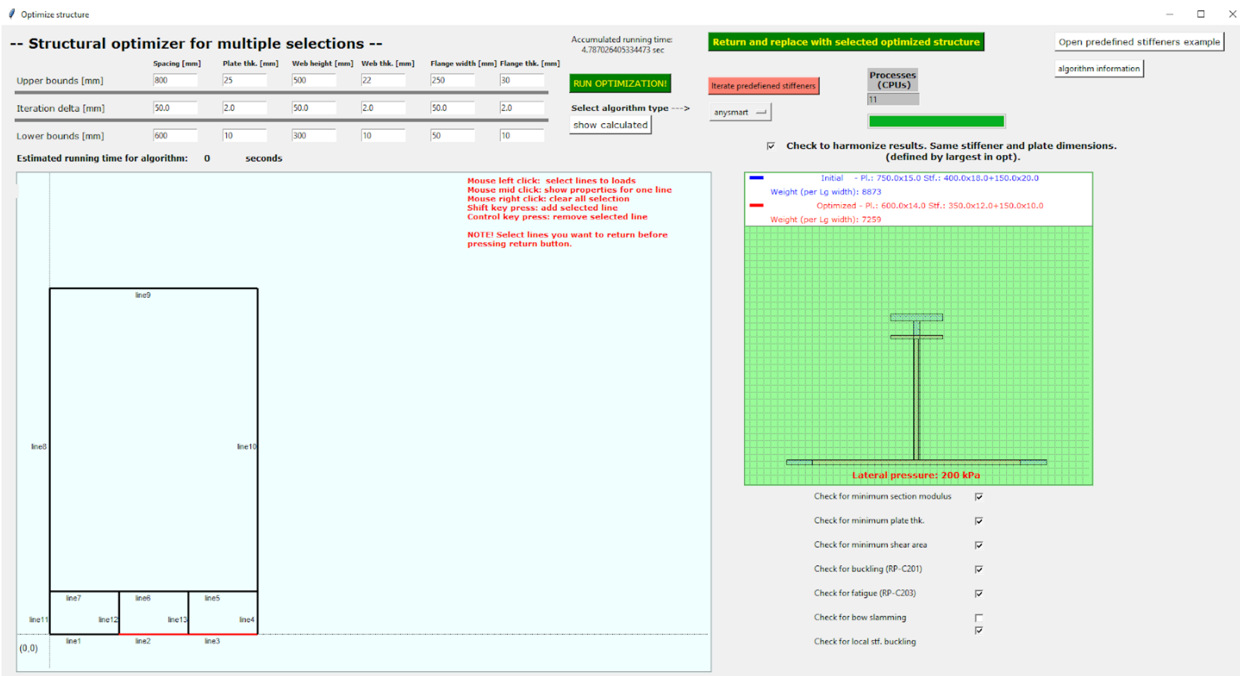


If you press the "show calculated" button, you will get an overview of how many is ok and how many failed (and what criteria first failed). One "occurence" is a one checked plate/stiffener combination.



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.**

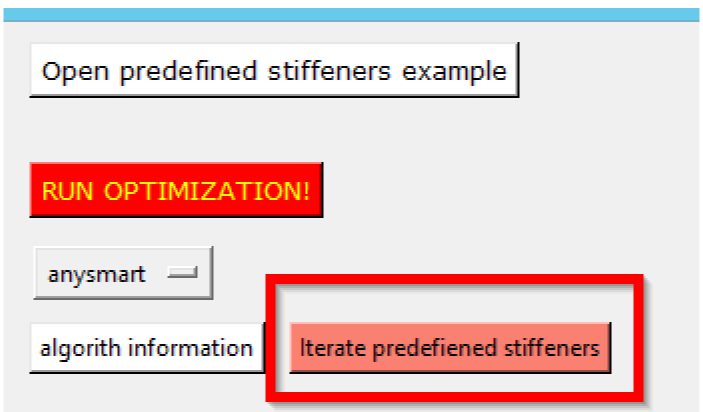
The optimization can be harmonized. That means that the largest dimension found in the multiple optimization is used for all selected. This is done after all

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).

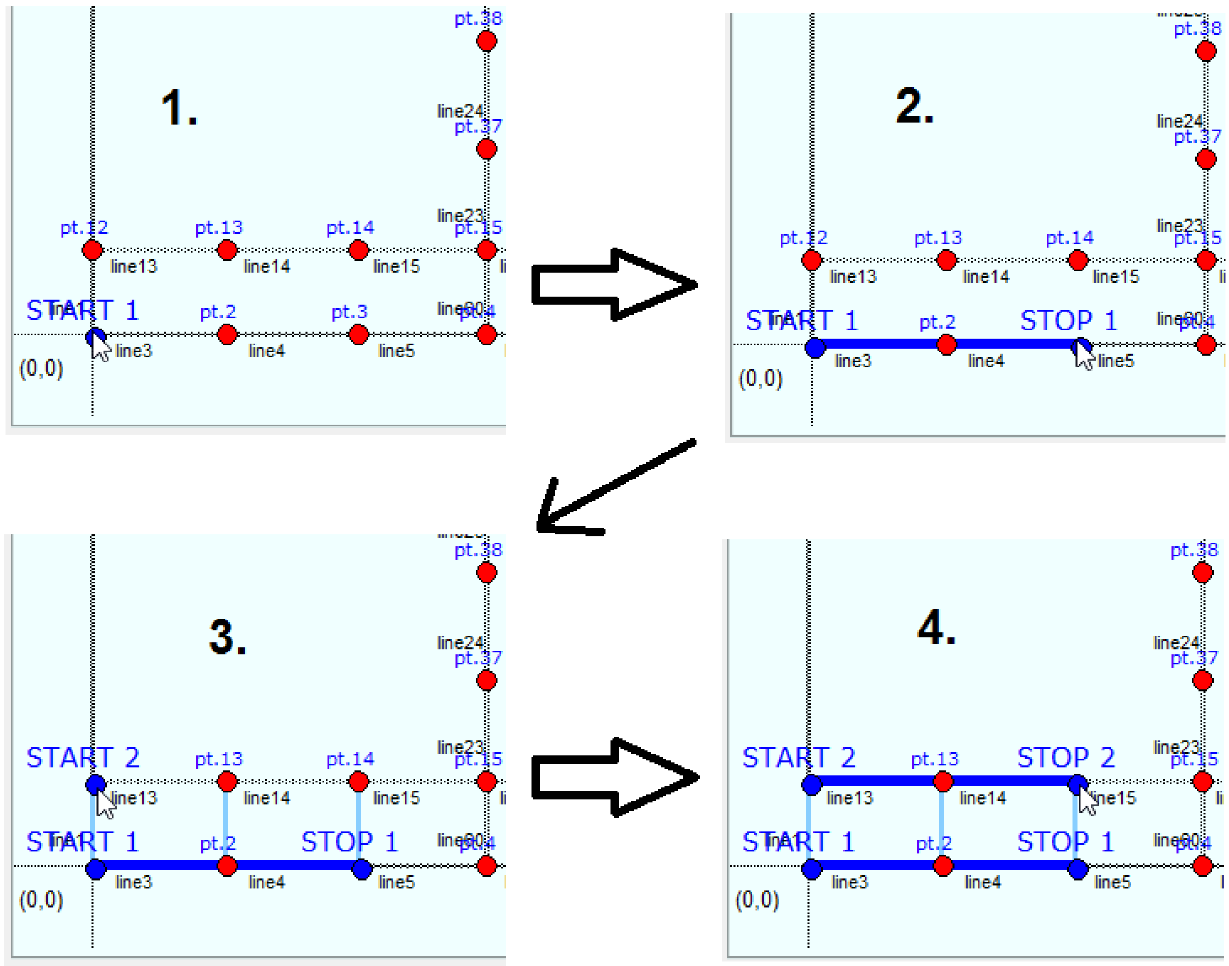
Span optimization

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



The optimization is started as follows.

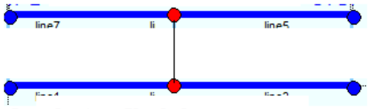
1. **Start by clicking as illustrated next:**



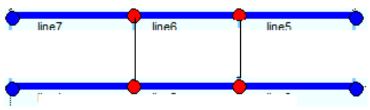
2. Then run optimization.

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.

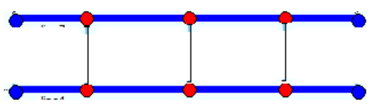
4 plate fields



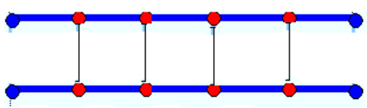
6 plate fields



8 plate fields



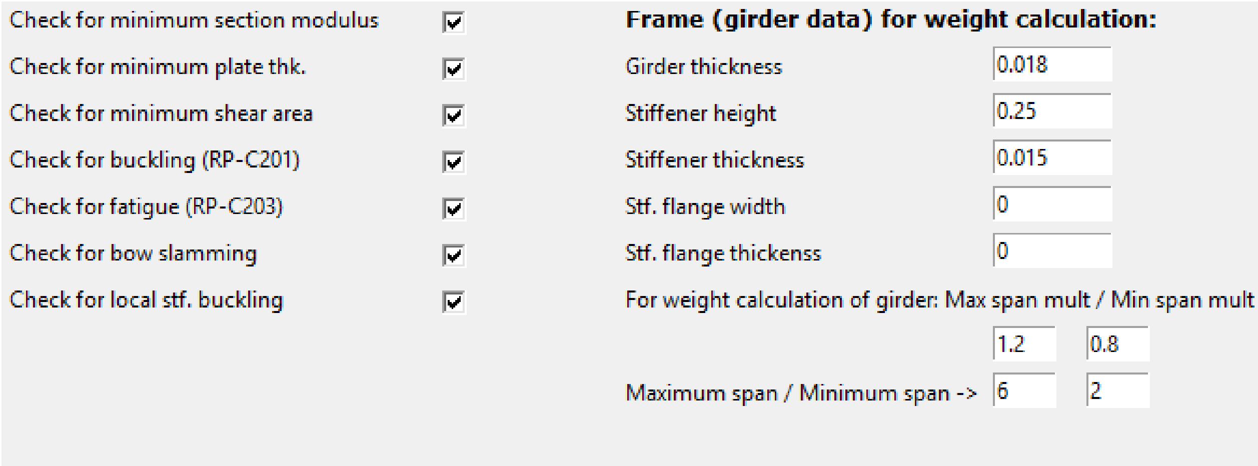
10 plate fields



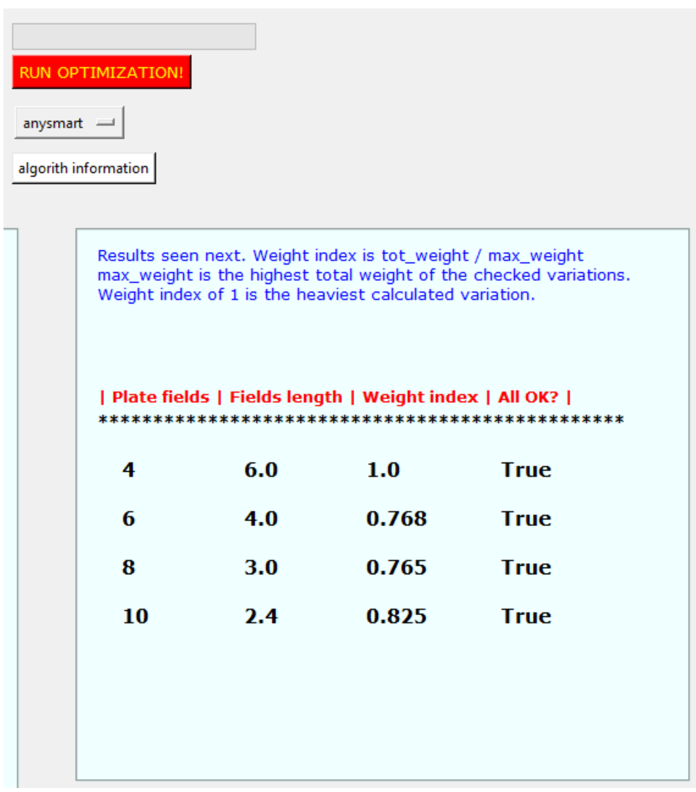
You can, similar to single optimization, select the checks that shall be runned. Also you can set the girder (frame) properties. This is used for calculating the weights.

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.

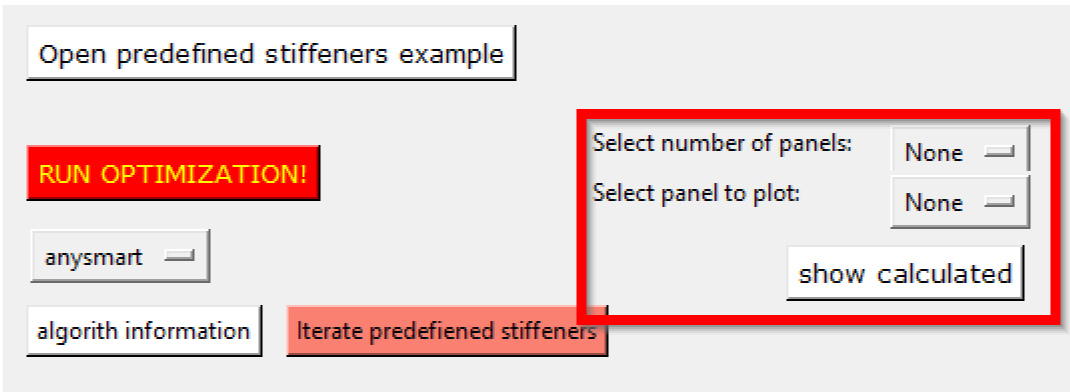


Results are presented as seen next.



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_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 ------- 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']
---------------------------------------------------------------------
```

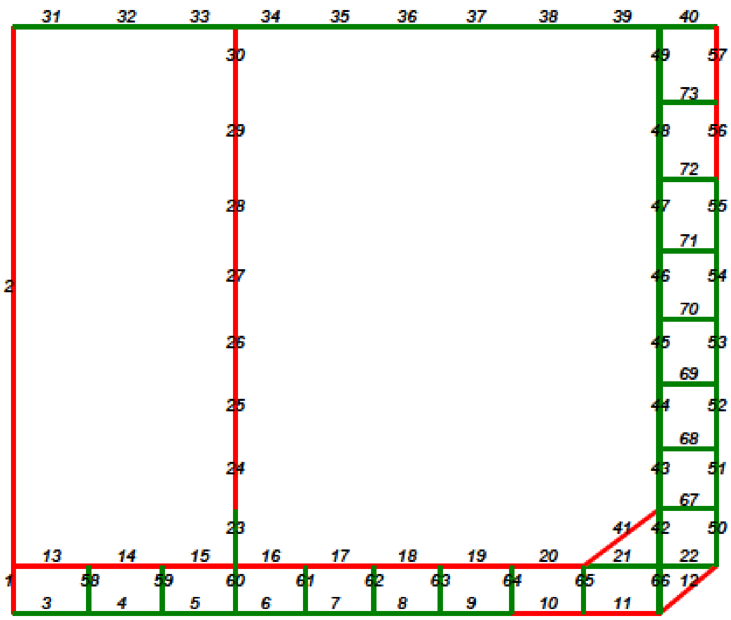
Reporting

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

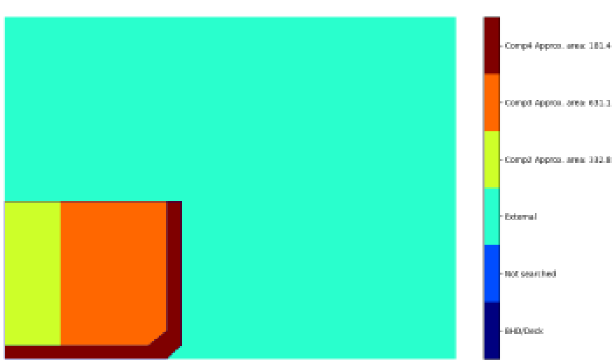
User: CEFANY

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



Compartments:

Compartments returned from search operation displayed below

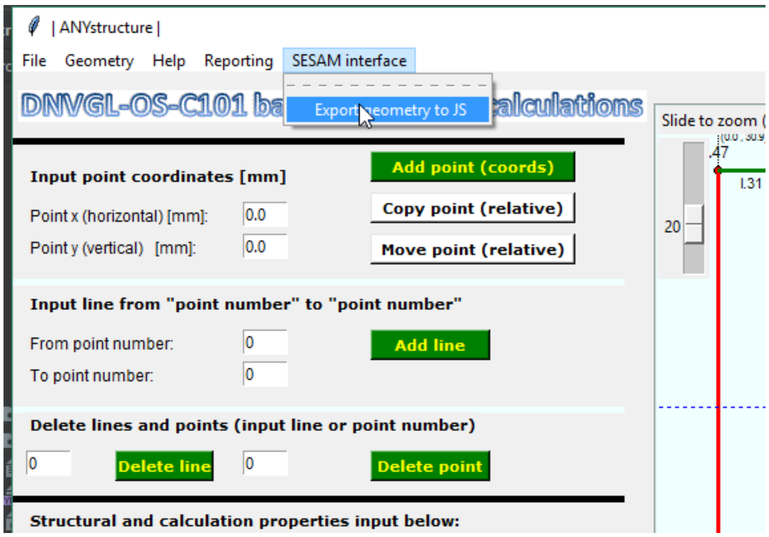


*area calculation inaccuracies due to thickness of barriers (BHD/Deck)

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)

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: