



File Geometry Reporting SESAM interface Help

Example file for ANYstructure: ship_section_example.txt

Input point coordinates [mm]

Point x (horizontal) [mm]: 0.0
Point y (vertical) [mm]: 0.0

Input line from "point number" to "point number"

From point number: 0
To point number: 0

Delete lines and points (or left/right click and use "Delete key")

Line number (left click): 10
Point number (right click): 0

Structural and calculation properties input below:

span: 3.8 [m]
s: 750.0 [mm]
p_thk: 18.0 [mm]
web_h: 400.0 [mm]
web_thk: 12.0 [mm]
fl_w: 150.0 [mm]
fl_thk: 20.0 [mm]
kpp: 1.0 [mm]
kps: 1.0 [mm]
sig_y1: 100.0 [MPa]
sig_y2: 100.0 [MPa]
sig_x: 50.0 [MPa]
tau_y1: 5.0 [MPa]
tau_y2: 5.0 [MPa]
stiffener type: T

Select structure type: BOTTOM

PULS input: SP Unst. ol.: UP
UP sup. left/right upper/lower
PULS acceptance: ultimate
PULS utilization factor: 0.87
Int-integrated GL-free left /right GT-free top/bottom
Continuous or Sniped: C

RP-C201 | PULS | ML
3
Run PULS update results

Find compartments

Comp. no.: 2
Tank content: ballast
Tank density: 0 [kg/m³]
Overpressure: 25000 [Pa]
Max elevation: 0.0
Min elevation: 0.0
Acceleration [m/s²]:

Display current compartments
Set compartment properties.
Delete all tanks

Check to see available shortcuts
Color coding
Beam prop.
Plate thk.
Pressure
Buckling UF
Sec. mod. UF
Fatigue UF
Total UF

Color coding
0.0
0.1
0.2
0.3
0.4
0.5
0.6
0.7
0.8
0.9
1.0

Static and dynamic accelerations

Static acceleration [m/s²]: 9.81
Dyn. acc. loaded [m/s²]: 3.0
Dyn. acc. ballast [m/s²]: 3.0

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
TankTest
Compartment4
Manual (pressure/LF)

Stat. LF 0.0 1.3 1.3 0.0 0.0 1.2 0.0
Dyn. LF 0.7 0.0 0.0 0.7 0.0 0.7 1.0
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.
DNV a [Pa]: [442744, 243755] DNV b [Pa]: [487211, 232562]
TT [Pa]: [295486] Manual [Pa]: [0.0]

PULS results for line **Load factors** **Load info**

Section moduluses: Wey1: 4.8700E+06 [mm³], Wey2: 1.7400E+06 [mm³]
Minimum section modulus: 1.9837E+06 [mm³]
Shear area: 5.2560E+03 [mm²]
Minimum shear area: 3.9500E+03 [mm²]
Plate thickness: 18.0 [mm]
Minimum plate thickness: 15.4 [mm]
Buckling results ANYstructure ML algorithm:
Buckling: UF below or equal 0.87
Ultimate: UF below or equal 0.87
CSR requirements (stiffener): plate-Ok web-Ok web/flange ratio-Ok flange-Ok
Fatigue results (DNVGL-RP-C203):
Total damage (DFF not included): 0.006 | With DFF = 2.0 --> Damage: 0.012

Plate field span: 3.8 meters
Stiffener spacing: 750.0 mm
Plate thickness: 18.0 mm
Stiffener web height: 400.0 mm
Stiffener web thickness: 12.0 mm
Stiffener flange width: 150.0 mm
Stiffener flange thickness: 20.0 mm
Material yield: 355.0 MPa
Structure/stiffener type: BOTTOM/T
Dynamic load variable: x - horizontal
Plate fixation parameter/kpp: 1.0
Stiff. fixation parameter/kps: 1.0
Global stress, sig_y1/sig_y2: 100.0/100.0 MPa
Global stress, sig_x: 50.0 MPa
Global shear, tau_xy: 5.0 MPa
km1, km2, km3: 12.0/24.0/12.0
Pressure side (p-plate/s-stf): p

Applied static/dynamic loads: ballast_bottom loaded_static ballast_static loaded_bottom loaded_bottom_FLS ballast_bottom_FLS TankTest

Documentation

2021

Version 3.X

	2
Introduction	3
Theory	3
Modelling	4
Assigning properties	5
Scale stresses	6
Changing multiple properties	7
Display properties	8
Define tanks	9
COB and COG	11
Setting accelerations	12
Define external pressures	13
Load combinations	14
Changing load factors	15
Buckling	16
Prescriptive	17
PULS integration	17
ML buckling	20
Reviewing data	21
Color coding	21
Shifting coordinates for visual purposes	21
Loads	22
Thickness and beam properties	24
Global stresses (buckling) and structure types	24
Results	25
Optimization	26
Optimization iteration by predefined stiffeners	26
Single optimization	27
Multiple optimization	29
Span optimization	30
Reporting	35
General	35
Table	36
Export to JS	36

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)

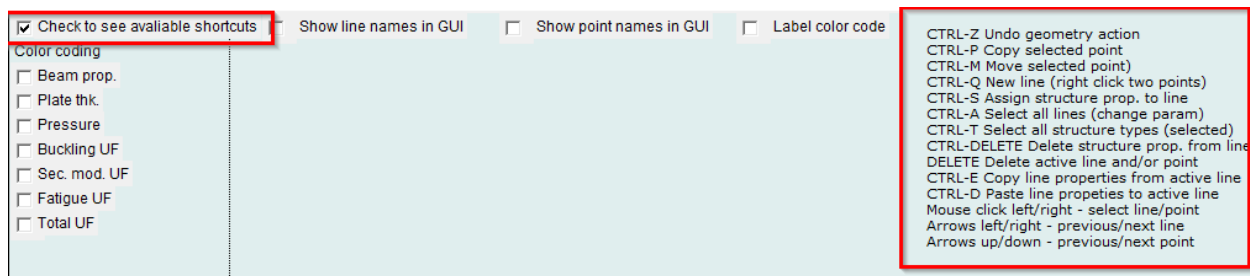
Input point coordinates [mm]		Add point (coords)
Point x (horizontal) [mm]:	<input type="text" value="0.0"/>	Copy point (relative)
Point y (vertical) [mm]:	<input type="text" value="0.0"/>	Move point (relative)
Input line from "point number" to "point number"		
From point number:	<input type="text" value="0"/>	Add line
To point number:	<input type="text" value="0"/>	
Delete lines and points (or left/right click and use "Delete key")		
Line number (left click):	<input type="text" value="43"/>	Delete line
Point number (right click):	<input type="text" value="0"/>	Delete point

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.

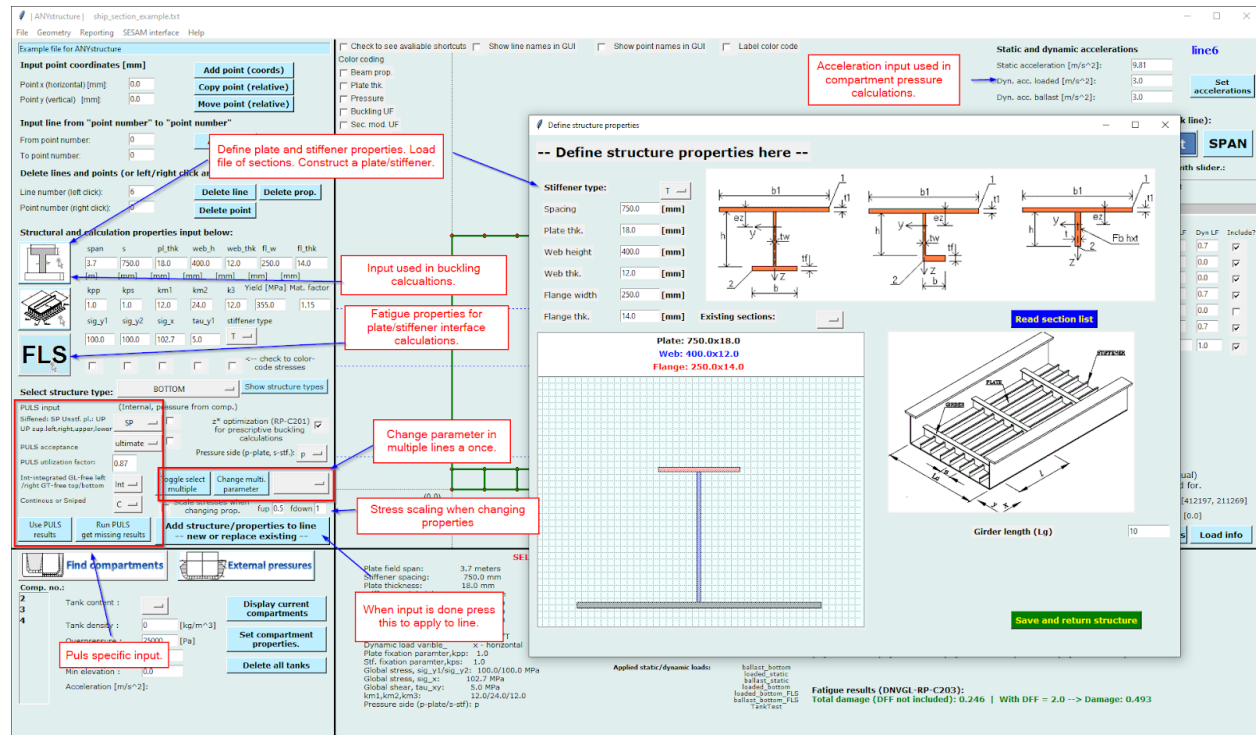


Assigning properties

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).

Structural and calculation properties input below:

	span	s	pl_thk	web_h	web_thk	fl_w	fl_thk
	2.4	800.0	20.0	200.0	15.0	0.0	0.0
	[m]	[mm]	[mm]	[mm]	[mm]	[mm]	[mm]

	kpp	kps	km1	km2	k3	Yield [MPa]	Mat. factor
	1.0	1.0	12.0	24.0	12.0	355.0	1.15

	sig_y1	sig_y2	sig_x	tau_y1	stiffener type
	10.0	10.0	180.0	4.0	FB

☐ ☐ ☐ ☐ ☐ <-- check to color-code stresses

Select structure type:

PULS input (Internal, pressure from comp.)

Siffened: SP Unstf. pl.: UP ☐ SSSS z* optimization (RP-C201) for prescriptive buckling calculations ☒

UP sup.left,right,upper,lower

PULS acceptance

PULS utilization factor: Pressure side (p-plate, s-stf.):

Int-integrated GL-free left /right GT-free top/bottom

Continuous or Sniped

☐ Scale stresses when changing prop. fup fdown

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).

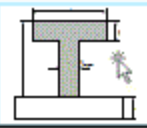
The parameters fdown = 1 and fup = 0.5 is by default. The general idea is that it is conservative to accept lower stress reduction when increasing thicknesses.

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.


Structural and calculation properties input below:



span	s	pl_thk	web_h	web_thk	fl_w	fl_thk
4	750	18	400	12	150	20
[m]	[mm]	[mm]	[mm]	[mm]	[mm]	[mm]



kpp	kps	km1	km2	k3	Yield [MPa]:
1	1	12	24	12	355



sig_y1	sig_y2	sig_x	tau_y1	stiffener type
90	90	40	5	T

☐ ☐ ☐ ☐ ☐ <-- check to color-code stresses

Select structure type: GENERAL_INTERNAL_WT Show structure types

PULS input (Internal, pressure from comp.)

Siffened: SP Unstf. pl.: UP sp z* optimization (RP-C201) for prescriptive buckling calculations ☒

UP sup.left,right,upper,lower ultimate

PULS acceptance 0.87 Pressure side (p-plate, s-stf.): p

PULS utilization factor: 0.87

Int-integrated GL-free left /right GT-free top/bottom Int

Continuous or Sniped C

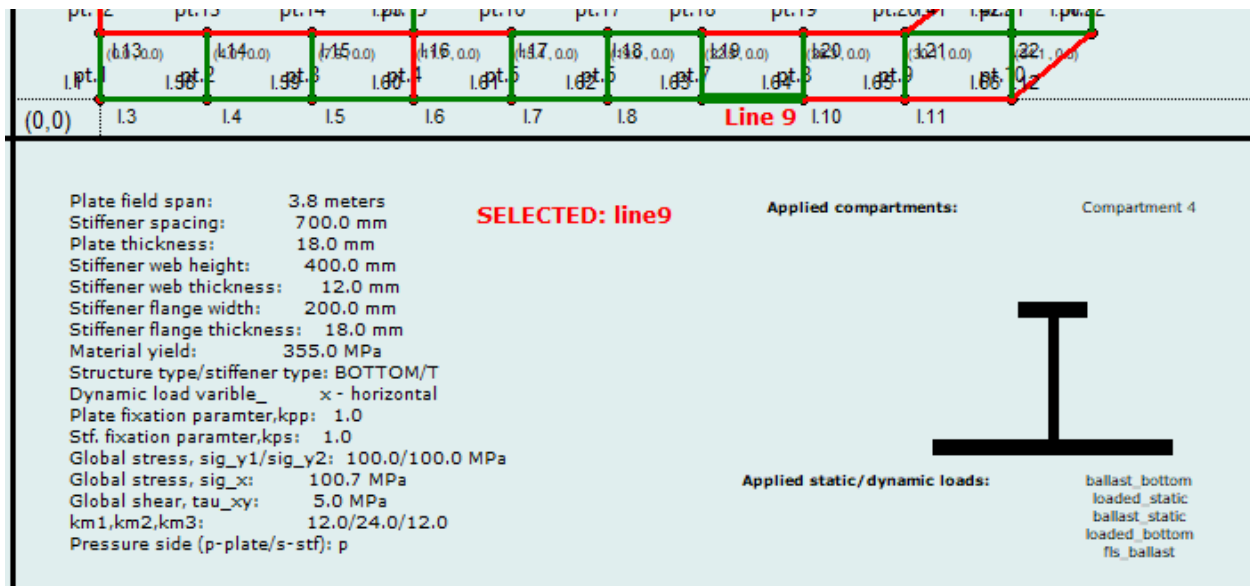
Toggle select multiple Change multi. parameter

☐ Scale stresses when changing prop.

Use PULS results
Run PULS get missing results
Add structure/properties to line -- new or replace existing --

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”.

By default tank content density is set to 1025 (water).

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

Tank density : 1000 [kg/m³]

Overpressure : 25000.0 [Pa]

Max elevation : 30.9

Min elevation : 2.5

Display current compartments

Set compartment properties.

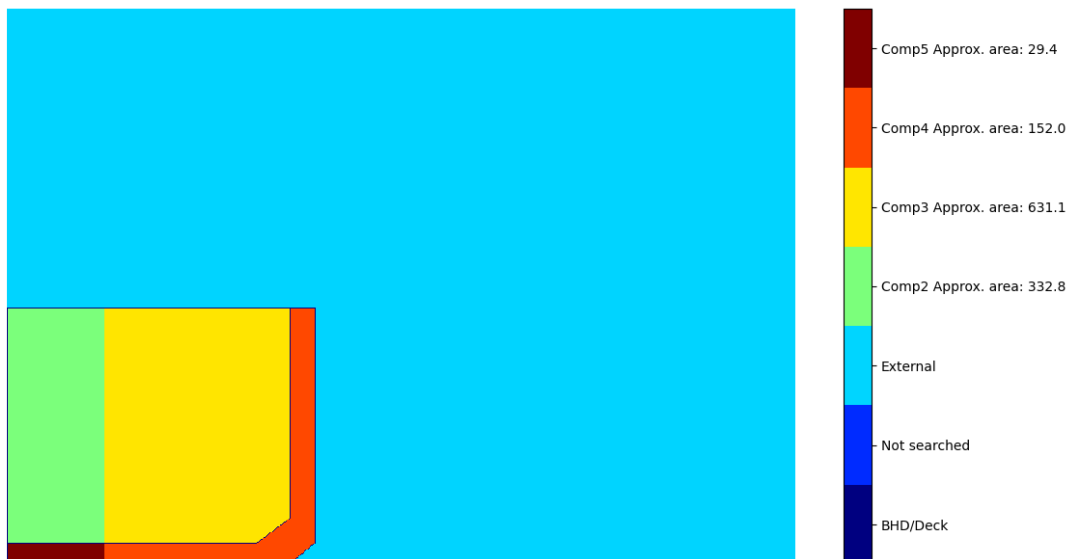
Delete all tanks

Accelerations [m/s²]:
 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.



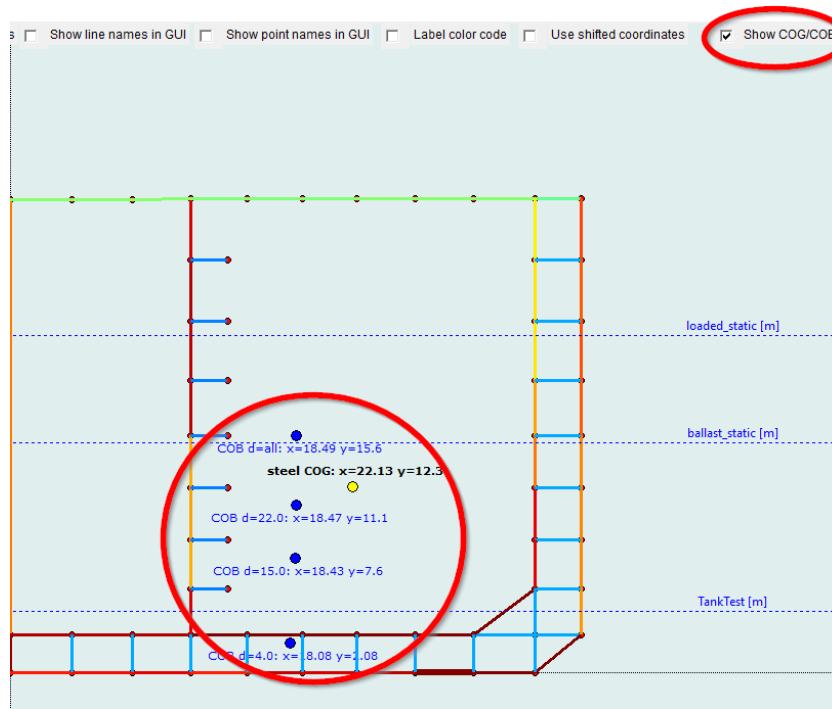
Compartments returned from search operation displayed below



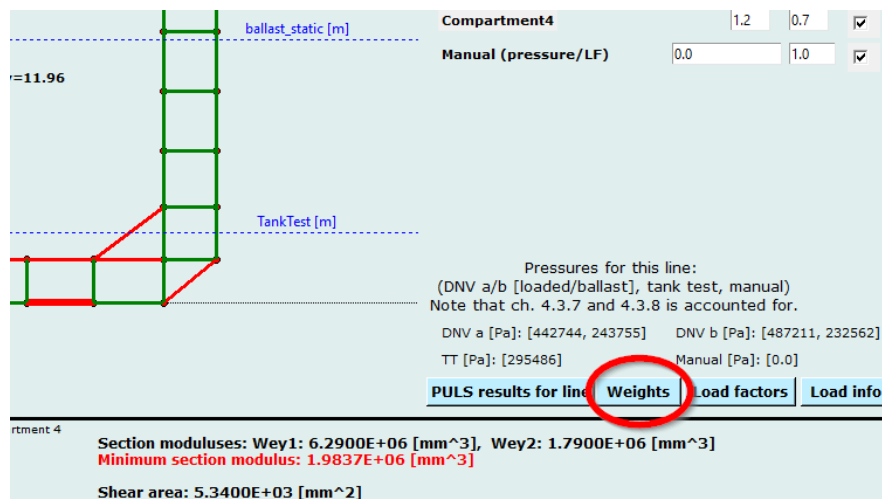
*area calculation inaccuracies due to thickness of barriers (BHD/Deck)

COB and COG

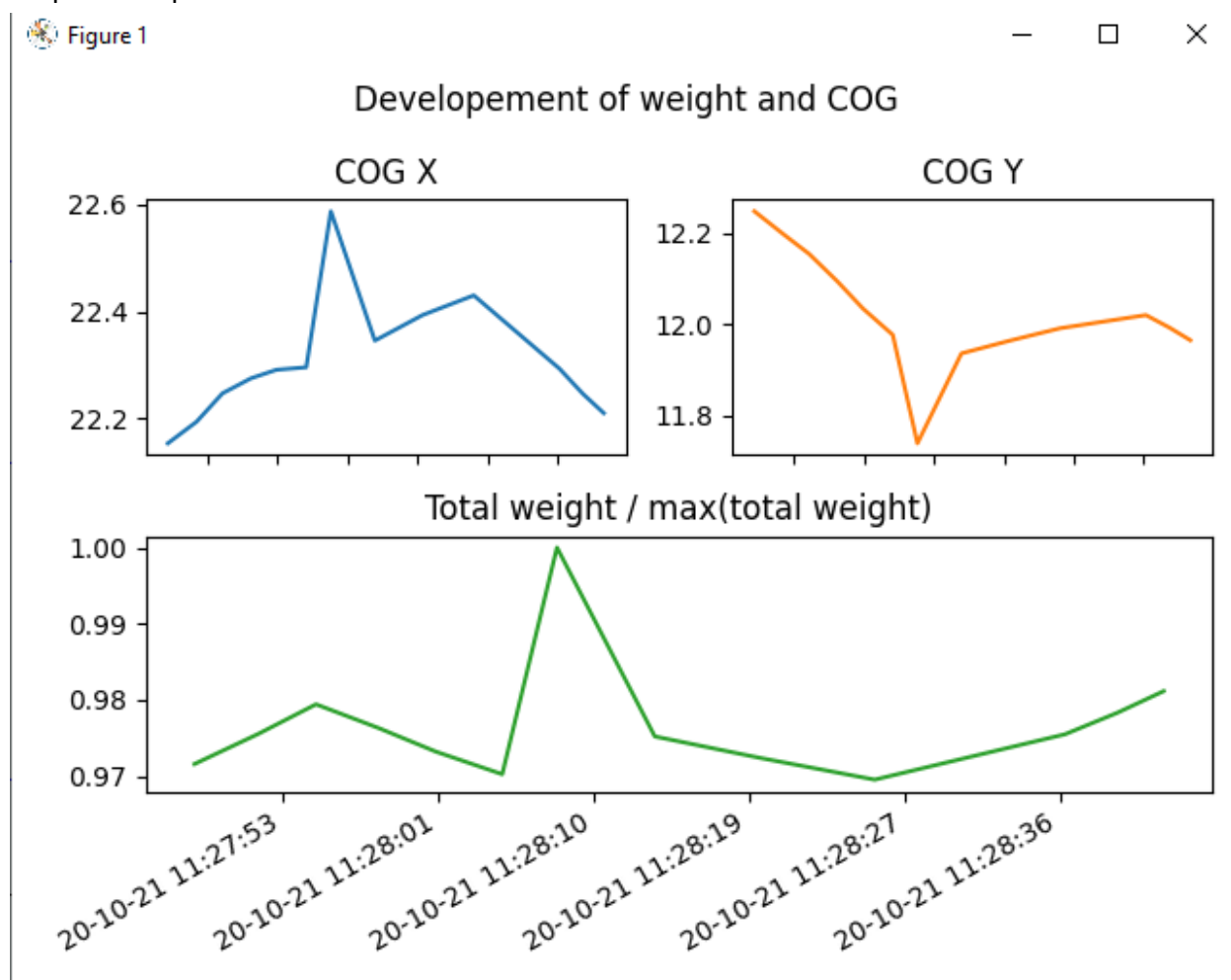
COG is calculated when structure properties are defined. COB for various drafts is calculated when tank search is completed.



Development of weight and COG is recorded each time the structure changes. The resulting plot can be seen by pressing the “weights” button.



Output example is seen next.



Setting accelerations

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

line10

Static and dynamic accelerations

Static acceleration [m/s ²]:	<input type="text" value="9.81"/>
Dyn. acc. loaded [m/s ²]:	<input type="text" value="3.0"/>
Dyn. acc. ballast [m/s ²]:	<input type="text" value="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 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]:

Plate multiplier, Ppl:

Stiffener multiplier, Pst:

Create slamming load

Press this to Save loads and close the load window.

Press to add selected lines to selecte 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

3. Created loads are seen below

(scroll if not all is shown.)
DOUBLE CLICK load to see associated lines.:

Select to see associated lines:

- static_22m
- slamming10bar
- loaded_bside_top
- loaded_side_shell
- loaded_above_wl
- ballast_bside_top
- ballast_side_shell
- ballast_above_wl
- static_15m
- static_8m_t
- ballast_bottom
- loaded_bottom_back
- loaded_bottom
- ballast_bottom_back
- FLS_ballast_bottom

-->

Delete selected load

Slamming loads can be specified separately. The multiply factors are the slamming pressure to be used on plate and stiffener respectively. For example for a 10 bar pressure:

$$P_{plate} = 10 \text{ bar} * \text{factor}_{plate}$$

$$P_{stf} = 10 \text{ bar} * \text{factor}_{stiffener}$$

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.

Combination for line (select line). Change with slider.:

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

1

Name:	Stat LF	Dyn LF	Include?
ballast_bottom	0.0	0.7	<input checked="" type="checkbox"/>
loaded_static	1.3	0.0	<input checked="" type="checkbox"/>
ballast_static	1.3	0.0	<input checked="" type="checkbox"/>
loaded_bottom	0.0	0.7	<input checked="" type="checkbox"/>
Compartment4	1.2	0.7	<input checked="" type="checkbox"/>
Manual (pressure/LF)	0.0	1.0	<input checked="" type="checkbox"/>

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.

DNV a [Pa]: [462698, 248632] DNV b [Pa]: [546435, 248430]
TT [Pa]: [335707] Manual [Pa]: [0.0]

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

Line:	Stat LF	Dyn LF	Include?
static_22m	1.3	0	<input checked="" type="checkbox"/>
static_15m	1.3	0	<input checked="" type="checkbox"/>
static_8m_tt	0	0	<input type="checkbox"/>
loaded_bottom	0	0.7	<input checked="" type="checkbox"/>
ballast_bottom	0	0.7	<input checked="" type="checkbox"/>
Compartment2	1.2	0.7	<input checked="" type="checkbox"/>
Manual (pressure/LF)	0	1	<input checked="" type="checkbox"/>

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.

DNV a [Pa]: [329265, 229422] DNV b [Pa]: [298631, 212755]
 TT [Pa]: [266326] Manual [Pa]: [0, 0]

Load factors **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 $\gamma_F = 1.0$ 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.

Buckling

ANYstructure has 3 options for calculating buckling.

Prescriptive

Buckling calculations as per DNV-RP-C203.

Structural and calculation properties input below:

span	s	pl_thk	web_h	web_thk	fl_w	fl_thk
3.8	750.0	18.0	400.0	12.0	150.0	20.0
[m]	[mm]	[mm]	[mm]	[mm]	[mm]	[mm]

kpp	kps	km1	km2	k3	Yield [MPa]	Mat. factor
1.0	1.0	12.0	24.0	12.0	355.0	1.15

sig_y1	sig_y2	sig_x	tau_y1	stiffener type
100.0	100.0	50.0	5.0	T

☐ ☐ ☐ ☐ ☐ <-- Color coding

Select structure type: BOTTOM

PULS input (Internal, pressure from comp.)

Siffened: SP Unstf. pl.: UP ☐ z* optimization (RP-C201) for buckling calculations ☒

UP sup.left.right.upper.lower

PULS acceptance ultimate ☐

PULS utilization factor: 0.87

Pressure side (p-plate, s-stf.): p

Int-integrated GL-free left /right GT-free top/bottom

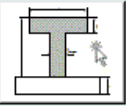


Continuous or Sniped ☐ Scale stresses when changing prop. fup 0.5 fdown 1

RP-C201 PULS | ML

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.

Structural and calculation properties input below:

  	span	s	pl_thk	web_h	web_thk	fl_w	fl_thk
	3.8	750.0	18.0	400.0	12.0	150.0	20.0
	[m]	[mm]	[mm]	[mm]	[mm]	[mm]	[mm]

kpp	kps	km1	km2	k3	Yield [MPa]	Mat. factor
1.0	1.0	12.0	24.0	12.0	355.0	1.15

sig_y1	sig_y2	sig_x	tau_y1	stiffener type
100.0	100.0	50.0	5.0	T

☐ ☐ ☐ ☐ ☐ <-- Color coding

Select structure type: BOTTOM Show structure types

PULS input (Internal, pressure from comp.)
 Stiffened: SP Unstf. pl.: UP sp ☐ z* optimization (RP-C201) for buckling calculations ☒
 UP sup.left.right.upper.lower ultimate ☐

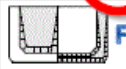

PULS acceptance Pressure side (p-plate, s-stf.): p

PULS utilization factor: 0.87

Int-integrated GL-free left /right GT-free top/bottom Int Toggle select multiple Change multi. parameter

Continuous or Sniped C ☐ Scale stresses when changing prop. fup 0.5 fdwn 1

RP-C201 | PULS | ML 2 Run PULS update results Add structure/properties to line -- new or replace existing --

 Find compartments  External pressures

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 consequently 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 and 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 %

```


ML buckling

The buckling ML option is an implementation of PULS using a neural network. It uses the same input as PULS.


The results should be used with care, as a neural network typically can give an incorrect prediction. The accuracy is currently in range of 97%.

ML buckling can be used for all optimization options.


Structural and calculation properties input below:



span	s	pl_thk	web_h	web_thk	fl_w	fl_thk
3.8	750.0	18.0	400.0	12.0	150.0	20.0
[m]	[mm]	[mm]	[mm]	[mm]	[mm]	[mm]



kpp	kps	km1	km2	k3	Yield [MPa]	Mat. factor
1.0	1.0	12.0	24.0	12.0	355.0	1.15



sig_y1	sig_y2	sig_x	tau_y1	stiffener type
100.0	100.0	50.0	5.0	T

☐ ☐ ☐ ☐ ☐ <-- Color coding

Select structure type: BOTTOM Show structure types

PULS input (Internal, pressure from comp.)

Siffened: SP Unstf. pl.: UP ☐ z* optimization (RP-C201) for buckling calculations ☒

UP sup.left,right,upper,lower ☐

PULS acceptance ultimate ☐

PULS utilization factor: 0.87

Int-integrated GL-free left /right GT-free top/bottom ☐ Int ☐

Continuous or Sniped ☐ C ☐

Pressure side (p-plate, s-stf.): p ☐

Toggle select multiple Change multi. parameter

Scale stresses when changing prop. fup 0.5 fdwn 1

RP-C201 | PULS | ML 3 Run PULS update results

Add structure/properties to line
-- new or replace existing --

Find compartments External pressures

Reviewing data

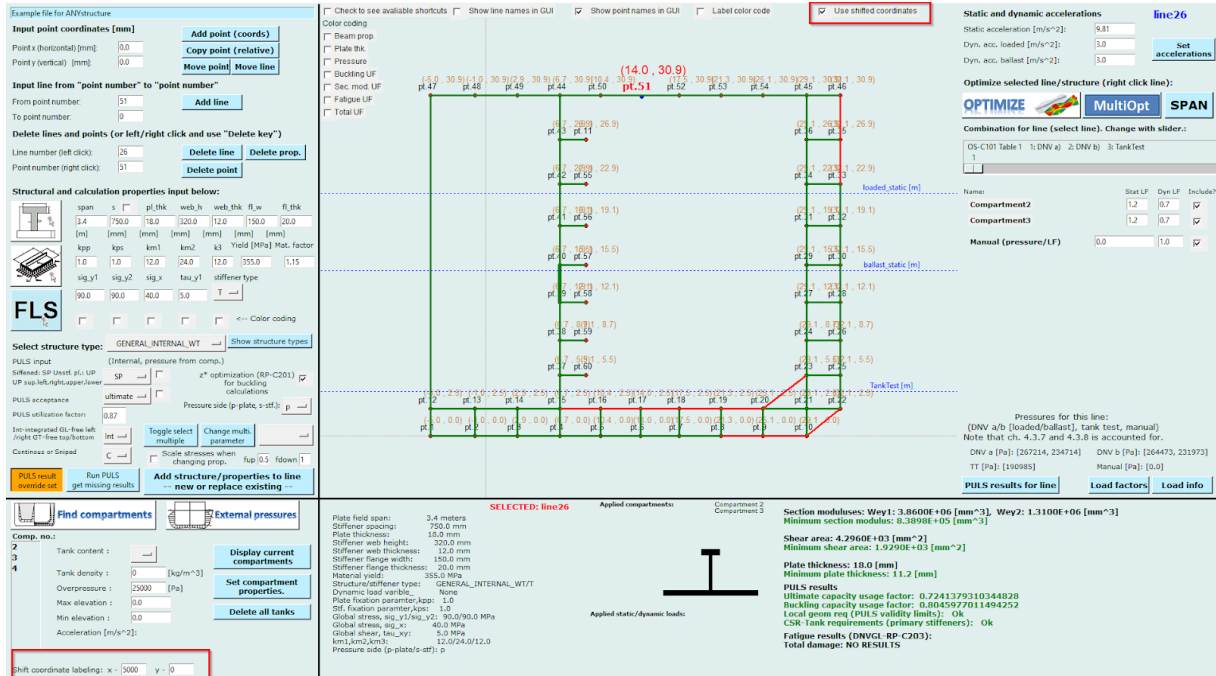
Color coding

All color coding options are indicated next.

The screenshot shows the ANSYS Workbench GUI for a ship section analysis. The 'Color coding' section is highlighted, showing options for 'Beam prop.', 'Plate thk.', 'Pressure', 'Buckling UP', 'Sec. mod UP', 'Fatigue UP', and 'Total UP'. The 'Structural and calculation properties input below' section is also visible, with a red box highlighting the 'Color coding' option. The main central area displays a 3D model of a ship's hull structure with various points and lines labeled. The 'Static and dynamic accelerations' panel on the right shows input values for static and dynamic accelerations. The bottom right panel displays 'PULS results for line' and 'Load factors'.

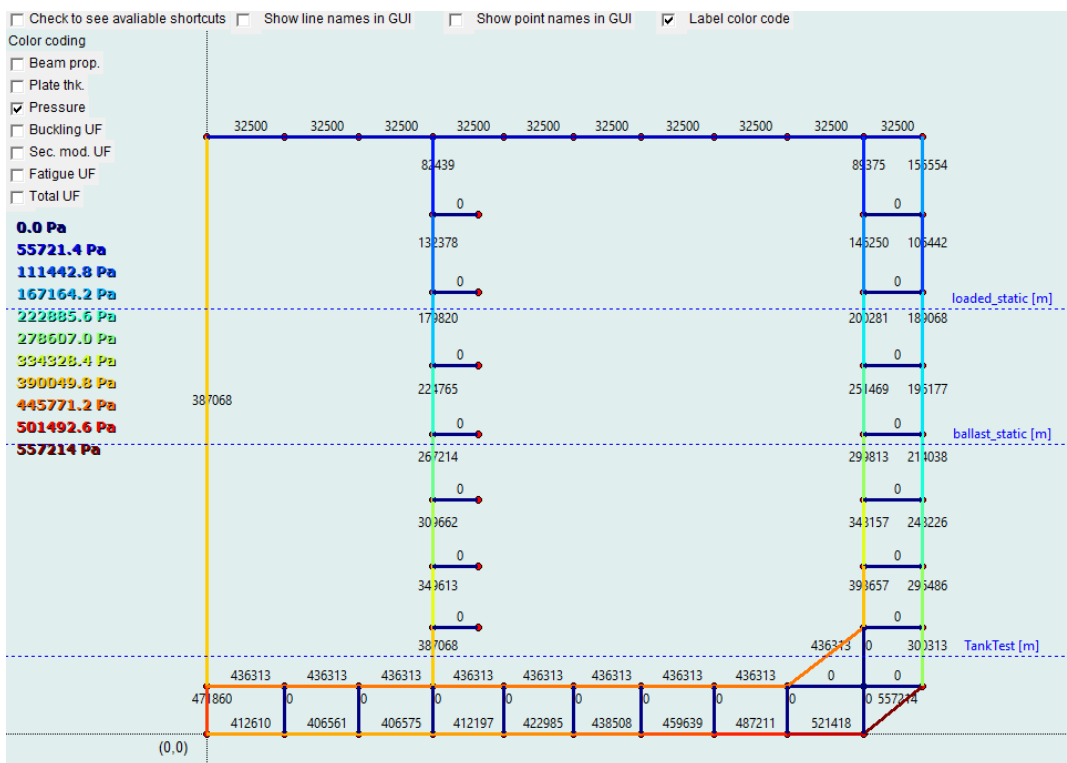
Shifting coordinates for visual purposes

You can shift the coordinates displayed for visual purposes. This does NOT affect the calculations of loads. This feature is included to better be able to review coordinates using different origins. Input magnitude of the shift in lower left corner and check “Use shifted coordinates” to activate.



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: $1 * 1.2 * 310707.225000000003 + 25000.0 * 1.3 = 405348.670000000004$

comp4 - dynamic: $1 * 0.7 * 95017.500000000001 + 25000.0 * 0 = 66512.25$

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: $1 * 1.0 * 310707.225000000003 + 25000.0 * 1.3 = 343207.225000000003$

comp4 - dynamic: $1 * 1.3 * 95017.500000000001 + 25000.0 * 0 = 123522.750000000003$

RESULT: $68910.2 + 150829 = 219739.0$

Tank test for: t

$1 * 1.0 * 40221.0 + 0 = 40221$

Tank test for: comp4

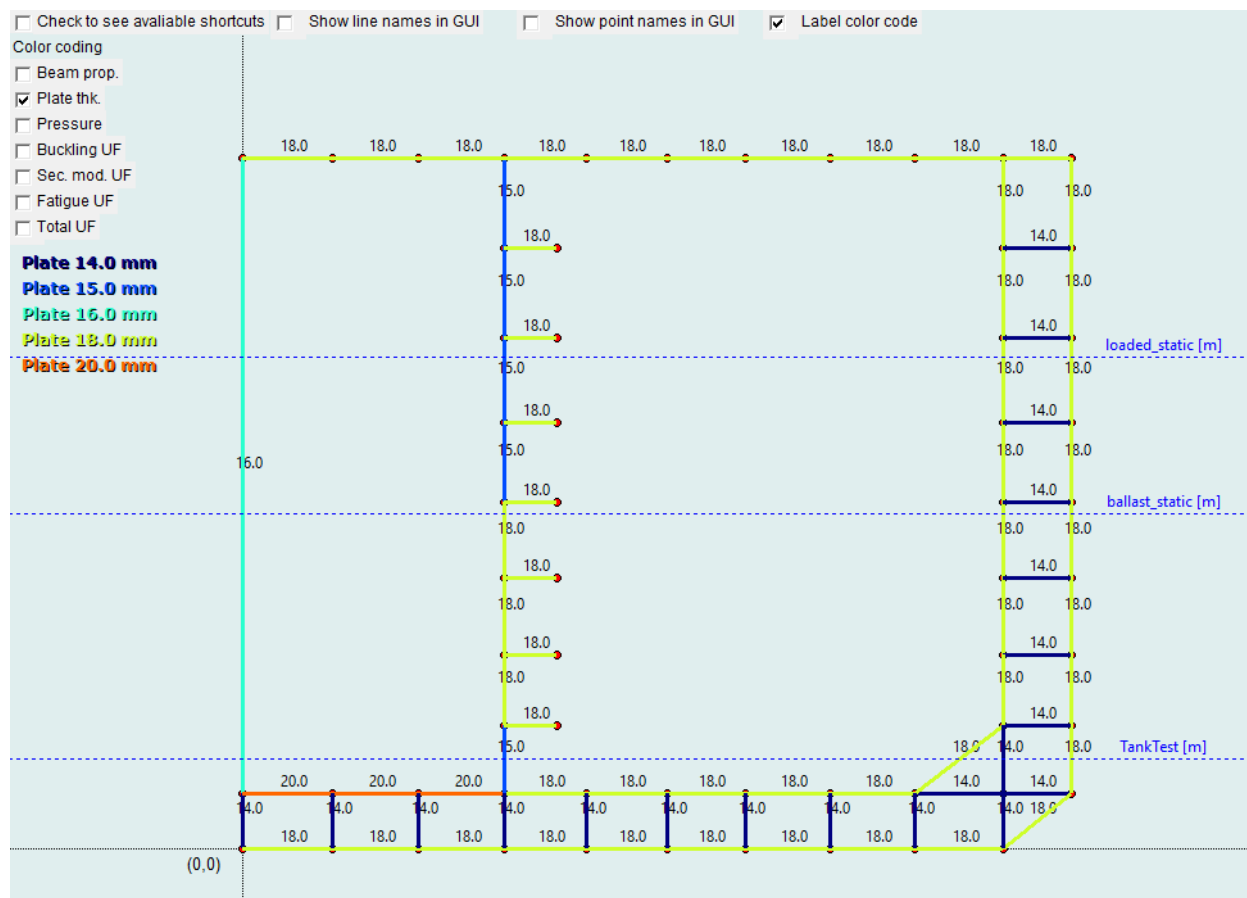
$1 * 1.0 * 310707.2 + 25000.0 * 1 = 335707$

Manual pressure:

$0.0 * 1.0 * 1 = 0.0$

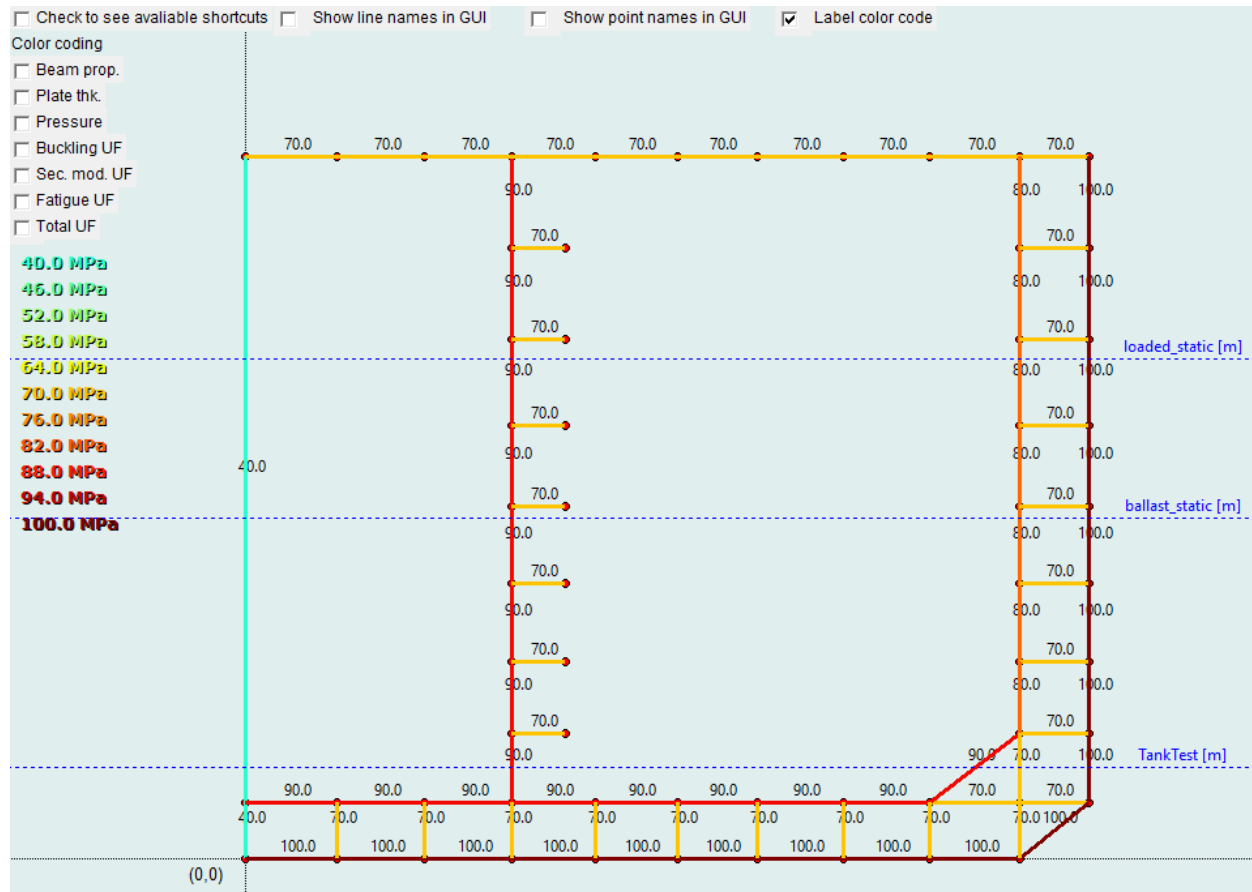
Thickness and beam properties

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 modulus not ok
Buckling not ok

<p>Section modulus: Wey1: 4.8300E+06 [mm^3], Wey2: 1.7500E+06 [mm^3] Minimum section modulus: 1.7163E+06 [mm^3]</p> <p>Shear area: 5.1600E+03 [mm^2] Minimum shear area: 3.5296E+03 [mm^2]</p> <p>Plate thickness: 18.0 [mm] Minimum plate thickness: 15.1 [mm]</p> <p>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</p> <p>Fatigue results (DNVGL-RP-C203): Total damage: NO RESULTS</p>	<p>Section modulus: Wey1: 4.2400E+06 [mm^3], Wey2: 1.4700E+06 [mm^3] Minimum section modulus: 2.0739E+06 [mm^3]</p> <p>Shear area: 4.6560E+03 [mm^2] Minimum shear area: 4.1297E+03 [mm^2]</p> <p>Plate thickness: 18.0 [mm] Minimum plate thickness: 15.8 [mm]</p> <p>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</p> <p>Fatigue results (DNVGL-RP-C203): Total damage (DFF not included): 0.058 With DFF = 2.0 --> Damage: 0.117</p>
--	---

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.

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

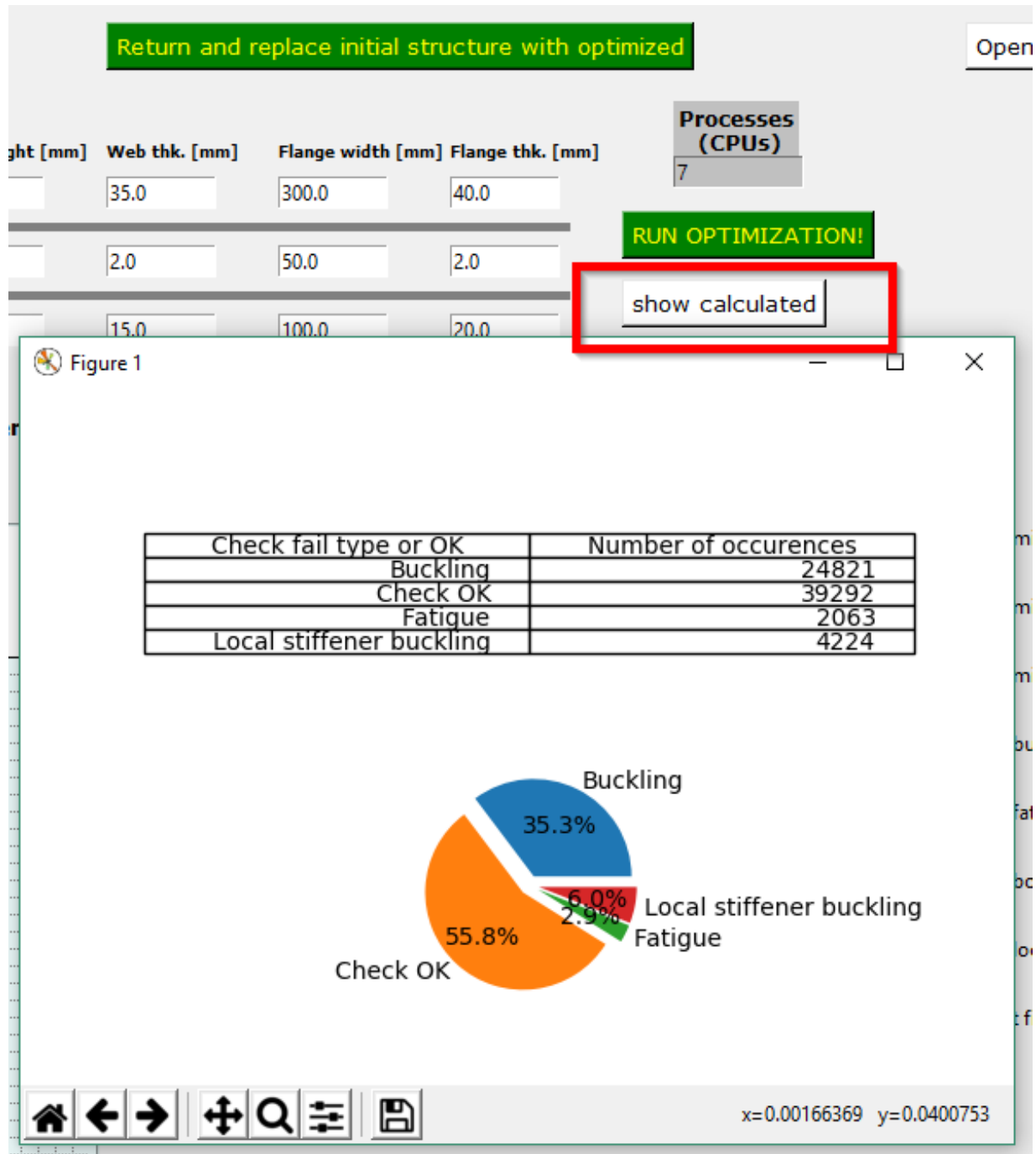
Various checks in the optimization module:

You can select the checks to be performed. 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.

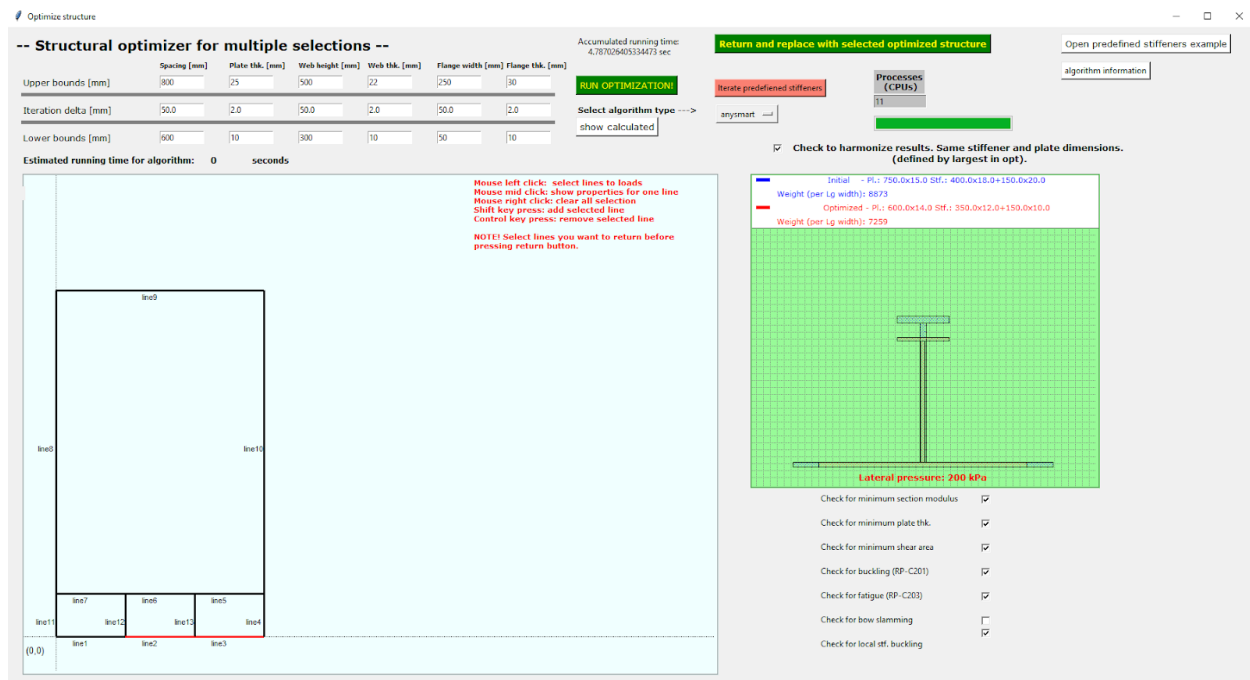
Check for minimum section modulus	<input checked="" type="checkbox"/>
Check for minimum plate thk.	<input checked="" type="checkbox"/>
Check for minimum shear area	<input checked="" type="checkbox"/>
Check for buckling (RP-C201)	<input checked="" type="checkbox"/>
Check for fatigue (RP-C203)	<input checked="" 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"/>
Check for buckling (PULS)	<input 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). One “occurrence” 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.

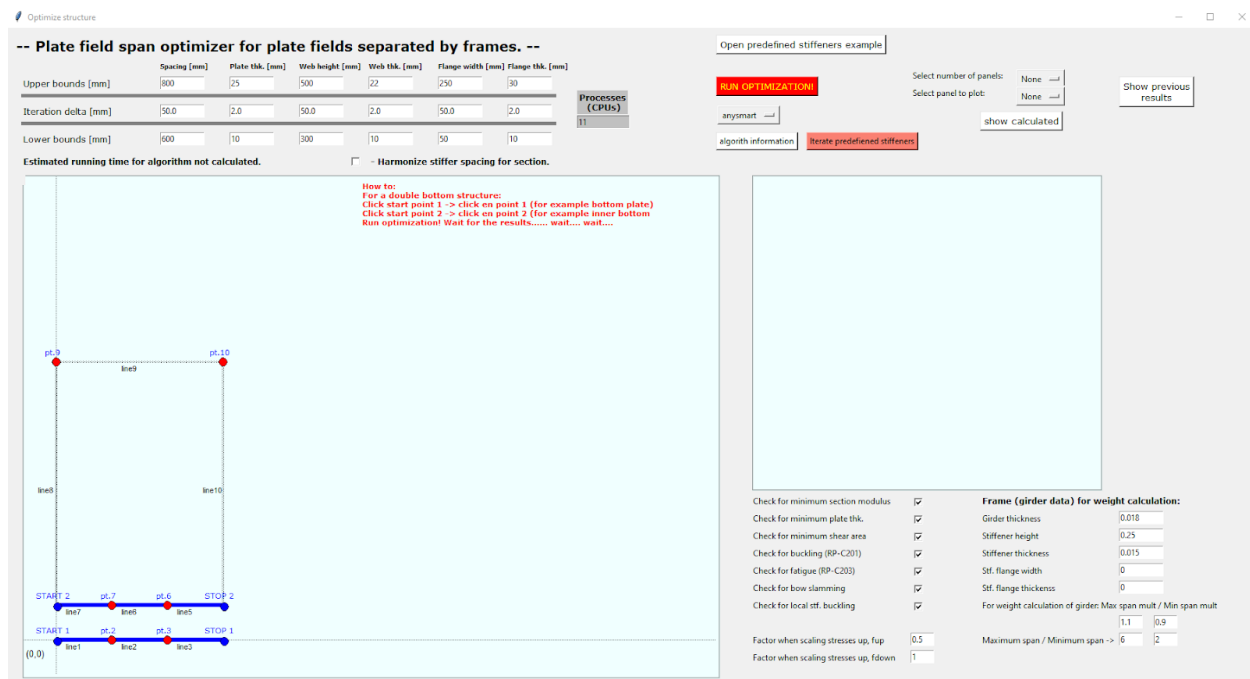
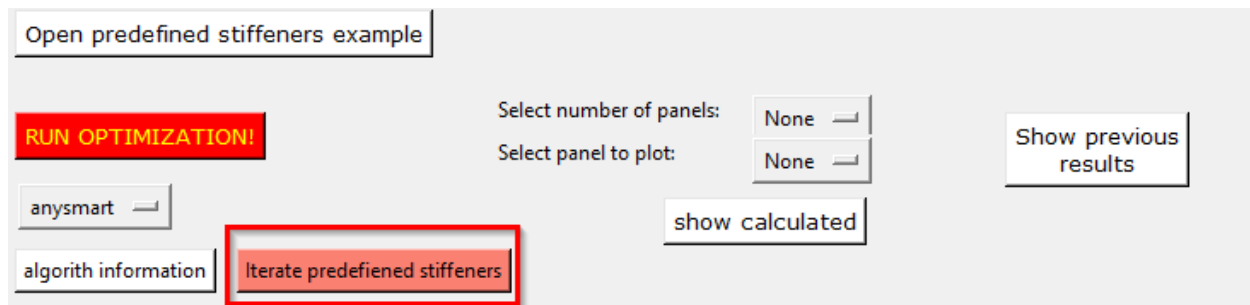
The optimization can be **harmonized**. That means that the largest dimension found in the multiple optimization is used for all selected. This is done after all

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).

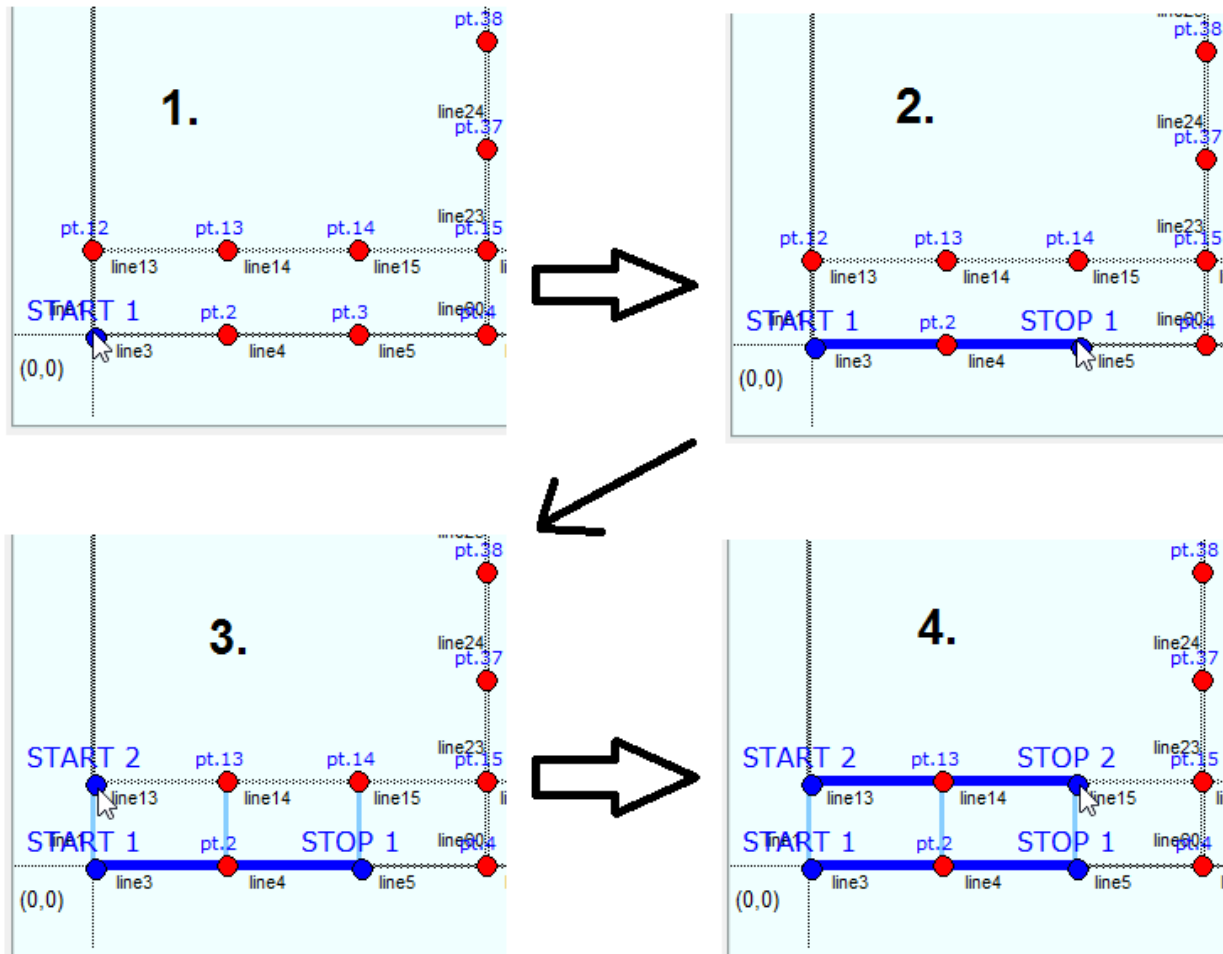
Span optimization

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. Check the input and checkboxes in the lower right corner

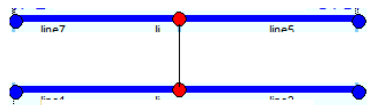
You can, similar to single optimization, select the checks that shall be runned. Also you can set the girder (frame) properties. This is used for calculating the weights.

Check for minimum section modulus	<input checked="" type="checkbox"/>	Frame (girder data) for weight calculation:	
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="1.1"/> <input type="text" value="0.9"/>
Factor when scaling stresses up, fup	<input type="text" value="0.5"/>	Maximum span / Minimum span ->	<input type="text" value="6"/> <input type="text" value="2"/>
Factor when scaling stresses up, fdwn	<input type="text" value="1"/>		

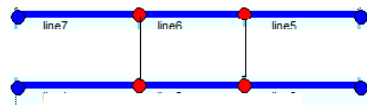
3. Start the calculation

The program will calculate variations of even spans in your structure as illustrated next. This is an example and the number of plate fields may vary.

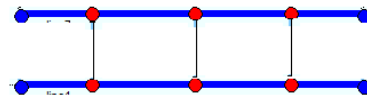
4 plate fields



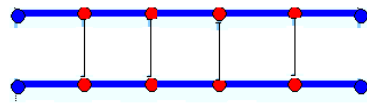
6 plate fields



8 plate fields



10 plate fields

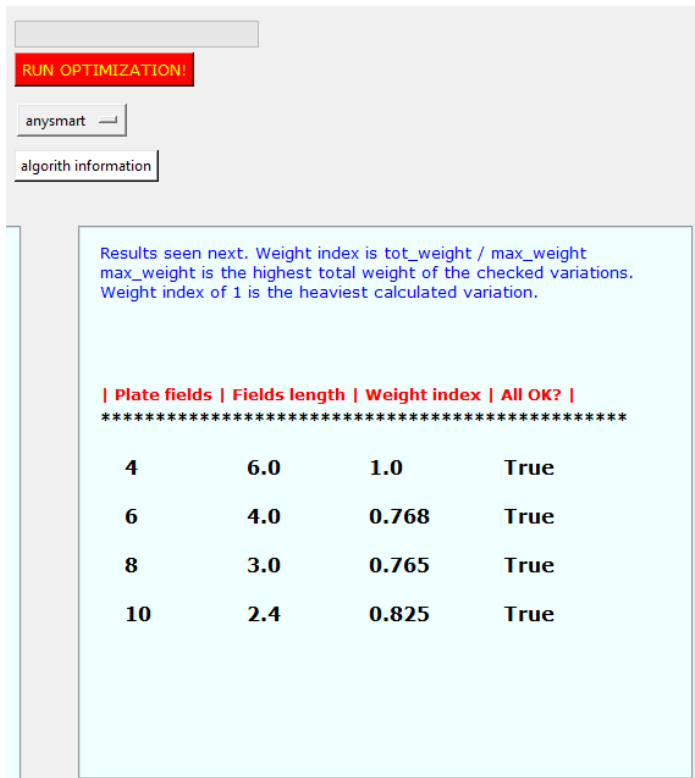


With reference to the previous example, max span mult is the multiplier for the 4 plate fields set up and min span mult is the weight multiplication for the 10 plate

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.

Results are presented as seen next.



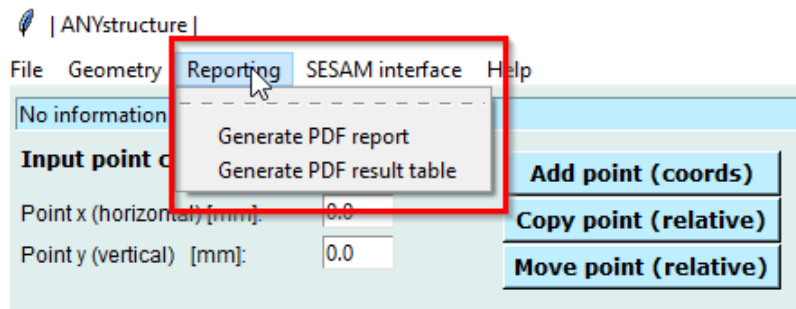
Results seen next. Weight index is $\text{tot_weight} / \text{max_weight}$
 max_weight is the highest total weight of the checked variations.
 Weight index of 1 is the heaviest calculated variation.

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 meters will give the lowest weight. 6 plate fields are 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,

Reporting



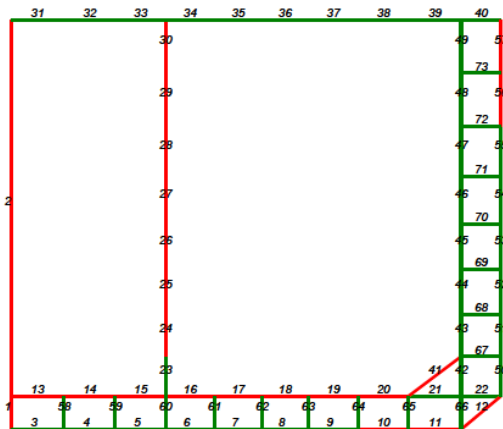
General

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
(p_stat, p_dyn_loa, p_dyn_hla): (9.81, 3.0, 3.0)
Name: comp3, content: crude_oil
Min. elevation: 2.5, Max. elevation: 30.900000000000002
Applied overpressure: 25000.0
(p_stat, p_dyn_loa, p_dyn_hla): (9.81, 3.0, 3.0)
Name: comp4, content: ballast
Min. elevation: 0.0, Max. elevation: 30.900000000000002
Applied overpressure: 25000.0
(p_stat, p_dyn_loa, p_dyn_hla): (9.81, 3.0, 3.0)

*Note calculation inaccuracies due to thickness of barriers (100/1000)

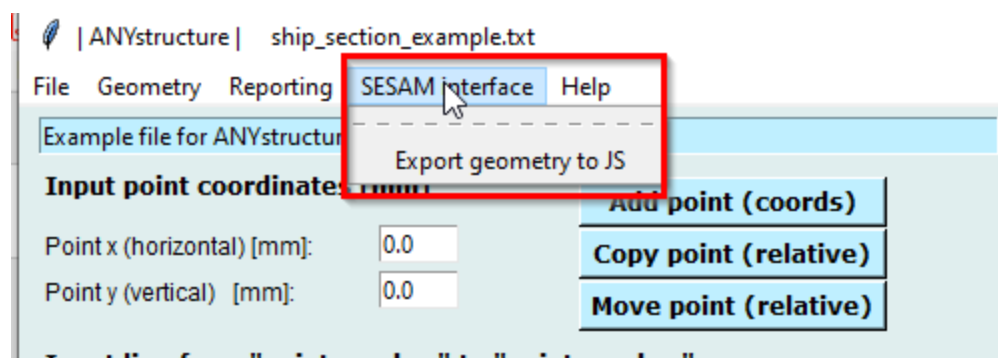
Table

A report in table format can be created using “Generate PDF result table”. This report is a compressed version of all results.

Line	pl thk	s	web h	web thk	fl. w	fl. thk	sig x	sig y1	sig y2	tau xy	max press.	sec. mod	min sec.	min plt	shr area	min shr A	fat uf	buc uf
line1	14.0	700.0	250.0	18.0	150.0	20.0	20.0	40.0	40.0	5.0	472000.0	1036266	630181	13.67	5111	2322		0.32
line10	18.0	750.0	400.0	12.0	150.0	20.0	50.0	100.0	100.0	5.0	487000.0	1737387	1983693	15.41	5256	3950	0.006	0.78
line11	18.0	750.0	500.0	12.0	150.0	20.0	50.0	100.0	100.0	5.0	521000.0	2318424	2352319	15.94	6456	4449	0.009	0.8
line12	18.0	750.0	500.0	12.0	150.0	20.0	50.0	100.0	100.0	5.0	557000.0	2318424	2395968	16.48	6456	4642	0.014	0.8
line13	20.0	775.0	450.0	12.0	150.0	20.0	40.0	90.0	90.0	5.0	436000.0	2044973	1959271	14.79	5880	3829		0.62
line14	20.0	775.0	450.0	12.0	150.0	20.0	40.0	90.0	90.0	5.0	436000.0	2044973	1959271	14.79	5880	3829		0.62
line15	20.0	775.0	450.0	12.0	150.0	20.0	40.0	90.0	90.0	5.0	436000.0	2044973	1959271	14.79	5880	3829		0.62
line16	18.0	775.0	375.0	12.0	150.0	18.0	40.0	90.0	90.0	5.0	436000.0	1499963	1676401	14.79	4932	3541		0.73
line17	18.0	775.0	375.0	12.0	150.0	18.0	40.0	90.0	90.0	5.0	436000.0	1499963	1587009	14.79	4932	3446		0.72
line18	18.0	775.0	375.0	12.0	150.0	18.0	40.0	90.0	90.0	5.0	436000.0	1499963	1500067	14.79	4932	3350		0.71
line19	18.0	775.0	375.0	12.0	150.0	18.0	40.0	90.0	90.0	5.0	436000.0	1499963	1768242	14.79	4932	3637		0.74
line2	16.0	700.0	400.0	18.0	150.0	20.0	20.0	40.0	40.0	5.0	387000.0	1931105	744392	12.38	7847	2286		0.27
line20	18.0	775.0	375.0	12.0	150.0	18.0	40.0	90.0	90.0	5.0	436000.0	1499963	1768242	14.79	4932	3637		0.74
line21	14.0	700.0	250.0	18.0	0.0	0.0	60.0	70.0	70.0	10.0	0.0	358911	3375	4.07	4751	0		0.23
line22	14.0	700.0	250.0	18.0	0.0	0.0	60.0	70.0	70.0	10.0	0.0	358911	3375	4.07	4751	0		0.22
line23	15.0	750.0	350.0	12.0	150.0	20.0	40.0	90.0	90.0	5.0	387000.0	1444282	946162	13.48	4620	2465		0.8
line24	18.0	750.0	350.0	12.0	150.0	20.0	40.0	90.0	90.0	5.0	350000.0	1468865	972352	12.81	4656	2375		0.63
line25	18.0	750.0	350.0	12.0	150.0	20.0	40.0	90.0	90.0	5.0	310000.0	1468865	972258	12.06	4656	2235		0.64
line26	18.0	750.0	320.0	12.0	150.0	20.0	40.0	90.0	90.0	5.0	267000.0	1314959	838982	11.2	4296	1929		0.63
line27	15.0	750.0	320.0	12.0	150.0	20.0	40.0	90.0	90.0	5.0	225000.0	1293077	791169	10.27	4260	1718		0.82
line28	15.0	750.0	320.0	12.0	150.0	20.0	40.0	90.0	90.0	5.0	180000.0	1293077	705247	9.19	4260	1450		0.82
line29	15.0	750.0	300.0	12.0	150.0	20.0	40.0	90.0	90.0	5.0	132000.0	1195190	575270	7.88	4020	1124		0.83
line3	18.0	700.0	400.0	12.0	200.0	20.0	102.0	100.0	100.0	5.0	413000.0	2109784	1856240	13.68	5256	3434	0.138	0.69
line30	15.0	750.0	300.0	12.0	150.0	20.0	40.0	90.0	90.0	5.0	82000.0	1195190	358252	6.22	4020	700		0.8

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 next:

