

File Geometry Reference SIMS interface Help
14/03/2024 ship_cmc11

Example file for ANVstructure

Input point coordinates [mm]

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

Input lines from "point number" to "point number"
 From point number: 0
 To point number: 0

Add line

Add point (coords)
Copy point (relative)
Move point (relative)

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

Line number (left click):	5	Delete line	Delete prop.
Point number (right click):	0	Delete point	

Structural and calculation properties input below:

span	3.7	sw_thk	web_h	web_thk	fl_w	fl_thk
ip1	750.0	18.0	40.0	12.0	250.0	14.0
ip2	100.0	18.0	40.0	12.0	250.0	14.0
ip3	100.0	18.0	40.0	12.0	250.0	14.0
ip4	100.0	18.0	40.0	12.0	250.0	14.0
ip5	100.0	18.0	40.0	12.0	250.0	14.0
ip6	100.0	18.0	40.0	12.0	250.0	14.0
ip7	100.0	18.0	40.0	12.0	250.0	14.0
ip8	100.0	18.0	40.0	12.0	250.0	14.0
ip9	100.0	18.0	40.0	12.0	250.0	14.0
ip10	100.0	18.0	40.0	12.0	250.0	14.0
ip11	100.0	18.0	40.0	12.0	250.0	14.0
ip12	100.0	18.0	40.0	12.0	250.0	14.0
ip13	100.0	18.0	40.0	12.0	250.0	14.0
ip14	100.0	18.0	40.0	12.0	250.0	14.0
ip15	100.0	18.0	40.0	12.0	250.0	14.0
ip16	100.0	18.0	40.0	12.0	250.0	14.0
ip17	100.0	18.0	40.0	12.0	250.0	14.0
ip18	100.0	18.0	40.0	12.0	250.0	14.0
ip19	100.0	18.0	40.0	12.0	250.0	14.0
ip20	100.0	18.0	40.0	12.0	250.0	14.0
ip21	100.0	18.0	40.0	12.0	250.0	14.0
ip22	100.0	18.0	40.0	12.0	250.0	14.0
ip23	100.0	18.0	40.0	12.0	250.0	14.0
ip24	100.0	18.0	40.0	12.0	250.0	14.0
ip25	100.0	18.0	40.0	12.0	250.0	14.0
ip26	100.0	18.0	40.0	12.0	250.0	14.0
ip27	100.0	18.0	40.0	12.0	250.0	14.0
ip28	100.0	18.0	40.0	12.0	250.0	14.0
ip29	100.0	18.0	40.0	12.0	250.0	14.0
ip30	100.0	18.0	40.0	12.0	250.0	14.0
ip31	100.0	18.0	40.0	12.0	250.0	14.0
ip32	100.0	18.0	40.0	12.0	250.0	14.0
ip33	100.0	18.0	40.0	12.0	250.0	14.0
ip34	100.0	18.0	40.0	12.0	250.0	14.0
ip35	100.0	18.0	40.0	12.0	250.0	14.0
ip36	100.0	18.0	40.0	12.0	250.0	14.0
ip37	100.0	18.0	40.0	12.0	250.0	14.0
ip38	100.0	18.0	40.0	12.0	250.0	14.0
ip39	100.0	18.0	40.0	12.0	250.0	14.0
ip40	100.0	18.0	40.0	12.0	250.0	14.0
ip41	100.0	18.0	40.0	12.0	250.0	14.0
ip42	100.0	18.0	40.0	12.0	250.0	14.0
ip43	100.0	18.0	40.0	12.0	250.0	14.0
ip44	100.0					

Version 2.X

Table of contents

Table of contents	2
Introduction	3
Theory	3
Modelling	4
Assigning properties	5
Scale stresses	7
Changing multiple properties	7
Display properties	8
Define tanks	9
Setting accelerations	11
Define external pressures	11
Load combinations	12
Changing load factors	13
PULS integration	14
Reviewing data	18
Loads	18
Thickness and beam properties	20
Global stresses (buckling) and structure types	20
Results	21
Optimization	22
Optimization iteration by predefined stiffeners	22
Single optimization	23
Multiple optimization	25
Span optimization	26
Reporting	31
Export to JS	32

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>



DNV·GL

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

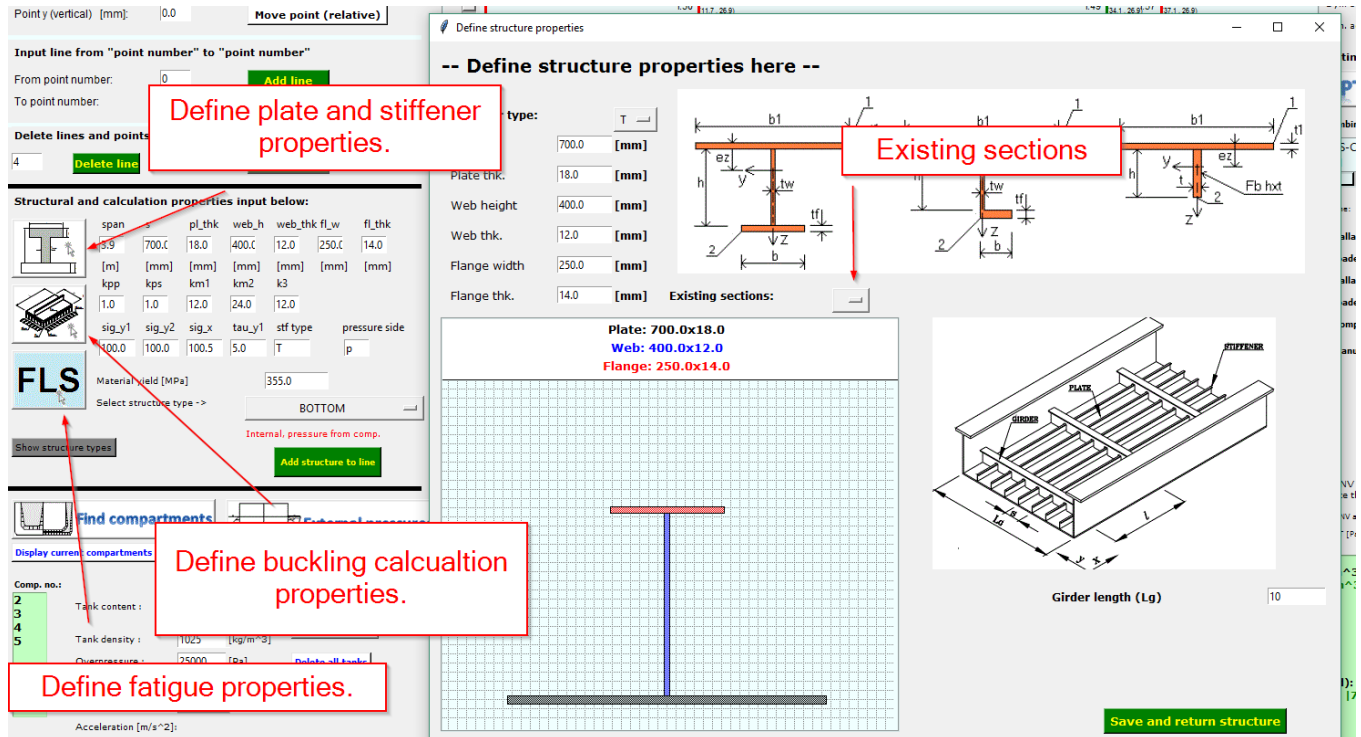
DELETE	Delete selected line/point
CTRL-E	Select a line and copy the properties of this line
CTRL-D	Paste structural properties to a selected line
Arrows up/down	Toggle point in model
Arrows left/right	Toggle lines in model
CTRL-A	Select all lines in model for changing a selected parameter for all
CTRL-T	Select all lines of a specific structure type for changing a parameter for multiple lines.

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).

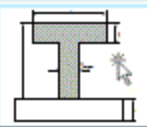
During automatic optimization fdown = 1 and fup = 0.5 by default. The general idea is that it is conservative to accept lower stress reduction when increasing thicknesses.

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

PULS utilization factor: 0.87

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

Continous or Sniped C

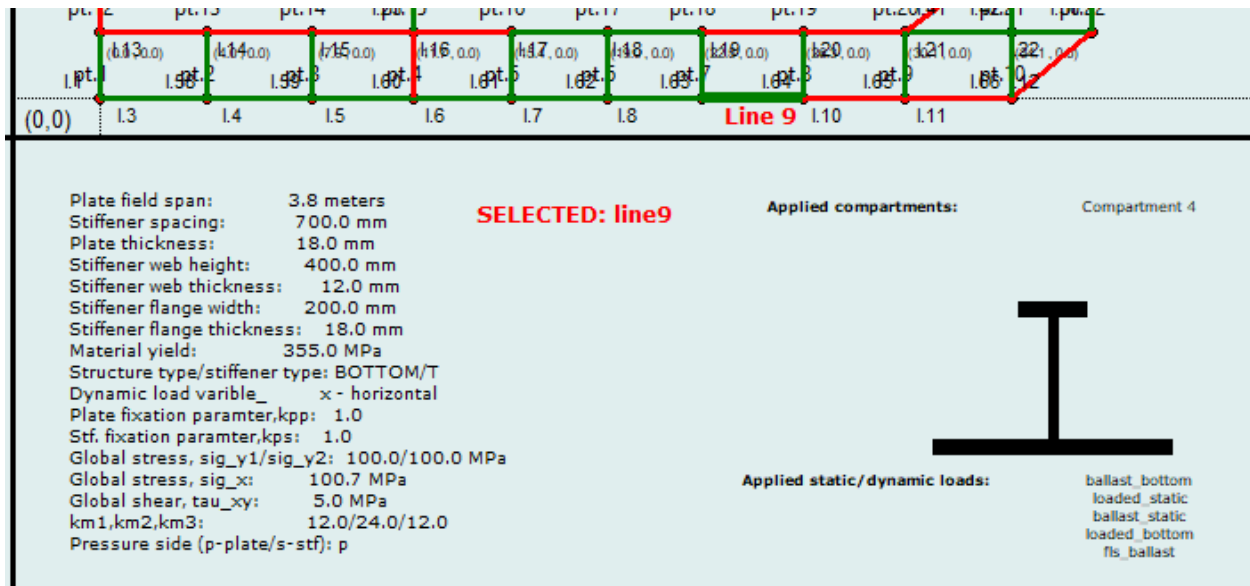
Toggle select multiple Change multi. parameter

☐ 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.




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/m³]

Set compartment properties.

Overpressure :

25000.0 [Pa]

Delete all tanks

Max elevation :

30.9

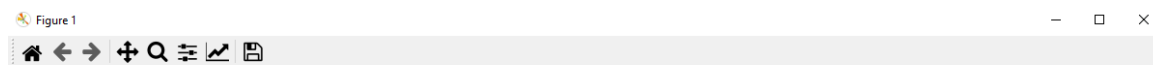
Min elevation :

2.5

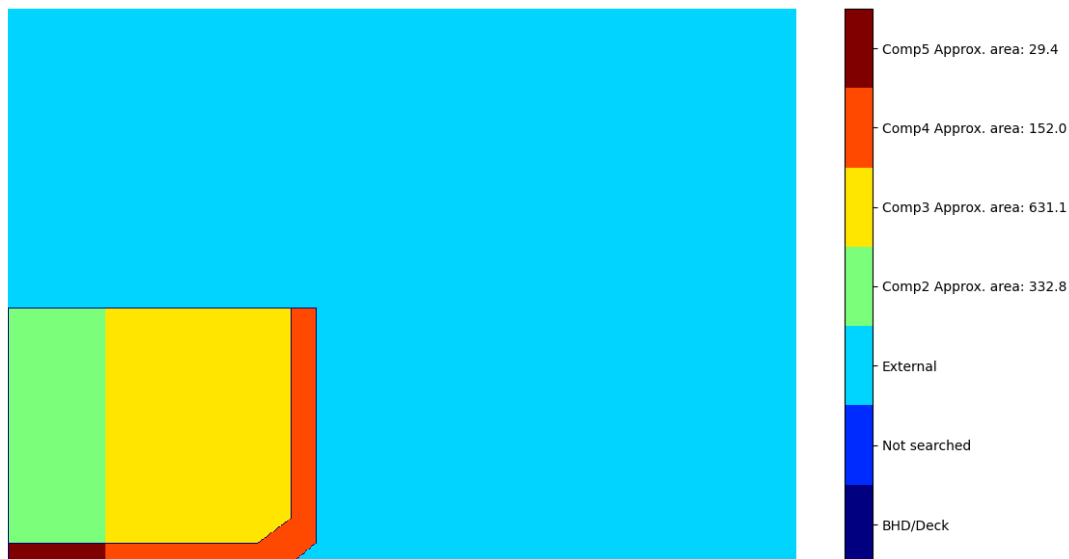
Accelerations [m/s²]:

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

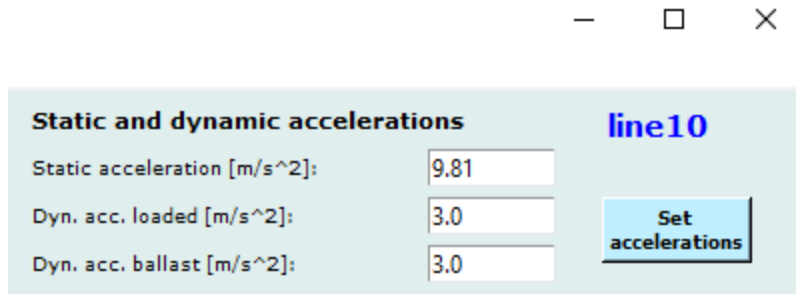


*area calculation inaccuracies due to thickness of barriers (BHD/Deck)



Setting accelerations

Accelerations applies to tank content. It 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
Dyn. acc. ballast [m/s ²]:	3.0

Set accelerations

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]:

Create slamming load

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

3. Created loads are seen below (scroll if not all is shown.)
DOUBLE CLICK load to see associated lines:

Select to see associated lines:

ballast_side	line50
ballast_bottom	line51
loaded_static	line52
ballast_static	line53
slamming	line54
loaded_bottom	line55
fls_ballast	

Delete selected load

Properties selected load is:

Name of load: ballast_side
Polynomial (x^3): 0.0
Polynomial (x^2): 303.0
Polynomial (x): -3750.0
Constant (C): 153000.0
Load condition: ballast
Limit state: ULS
Is external?: True
Static draft: None

Press to add selected lines to selected load

Select a load in "3." to and then choose lines to apply to load (select by clicking lines). Alternatively define manually ----->

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

line1 line2 line3 line4 line5 line6 line7 line8 line9 line10 line11 line12 line13 line14 line15 line16 line17 line18 line19 line20 line21 line22 line23 line24 line25 line26 line27 line28 line29 line30 line31 line32 line33 line34 line35 line36 line37 line38 line39 line40 line41 line42 line43 line44 line45 line46 line47 line48 line49 line50 line51 line52 line53 line54 line55 line56 line57 line58 line59 line60 line61 line62 line63 line64 line65 line66 line67

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.

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.

No information on project provided. Input here.

Input point coordinates [mm]

Point x (horizontal) [mm]:

Point y (vertical) [mm]:

Add point (coords)

Copy point (relative)

Move point (relative)

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

From point number:

To point number:

Add line

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

Line number (left click):

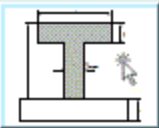


Point number (right click):

Delete line

Delete prop.

Delete point

Structural and calculation properties input below:

span	s	pl_thk	web_h	web_thk	fl_w	fl_thk
<input type="text" value="4"/>	<input type="text" value="750"/>	<input type="text" value="18"/>	<input type="text" value="400"/>	<input type="text" value="12"/>	<input type="text" value="150"/>	<input type="text" value="20"/>
[m]	[mm]	[mm]	[mm]	[mm]	[mm]	[mm]

kpp	kps	km1	km2	k3	Yield [MPa]:
<input type="text" value="1"/>	<input type="text" value="1"/>	<input type="text" value="12"/>	<input type="text" value="24"/>	<input type="text" value="12"/>	<input type="text" value="355"/>

sig_y1	sig_y2	sig_x	tau_y1	stiffener type
<input type="text" value="90"/>	<input type="text" value="90"/>	<input type="text" value="40"/>	<input type="text" value="5"/>	<input type="text" value="T"/>

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

Select structure type: **Show structure types**

PULS input (Internal, pressure from comp.)

Siffened: SP Unstf. pl.: UP

UP sup,left,right,upper,lower

PULS acceptance

PULS utilization factor:

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

Continuous or Sniped

z* optimization (RP-C201) for prescriptive buckling calculations ☒

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

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 --**

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:


```

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 %

```


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

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.

Section modulus not ok
Buckling not ok

<div>Section modulus: 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</div>	<div>Section modulus: 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</div>
---	--

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

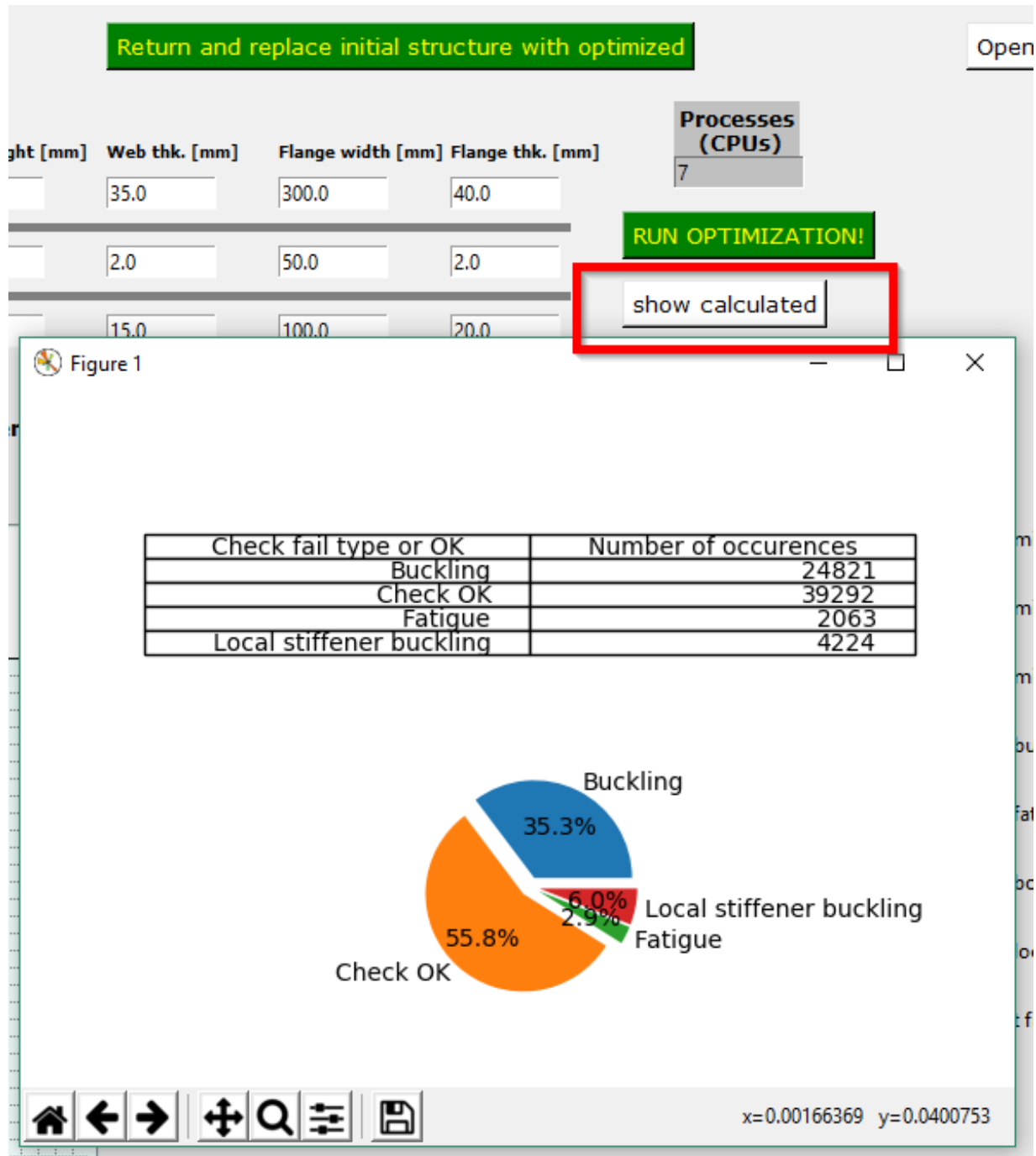
Various checks in the optimization module:

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.

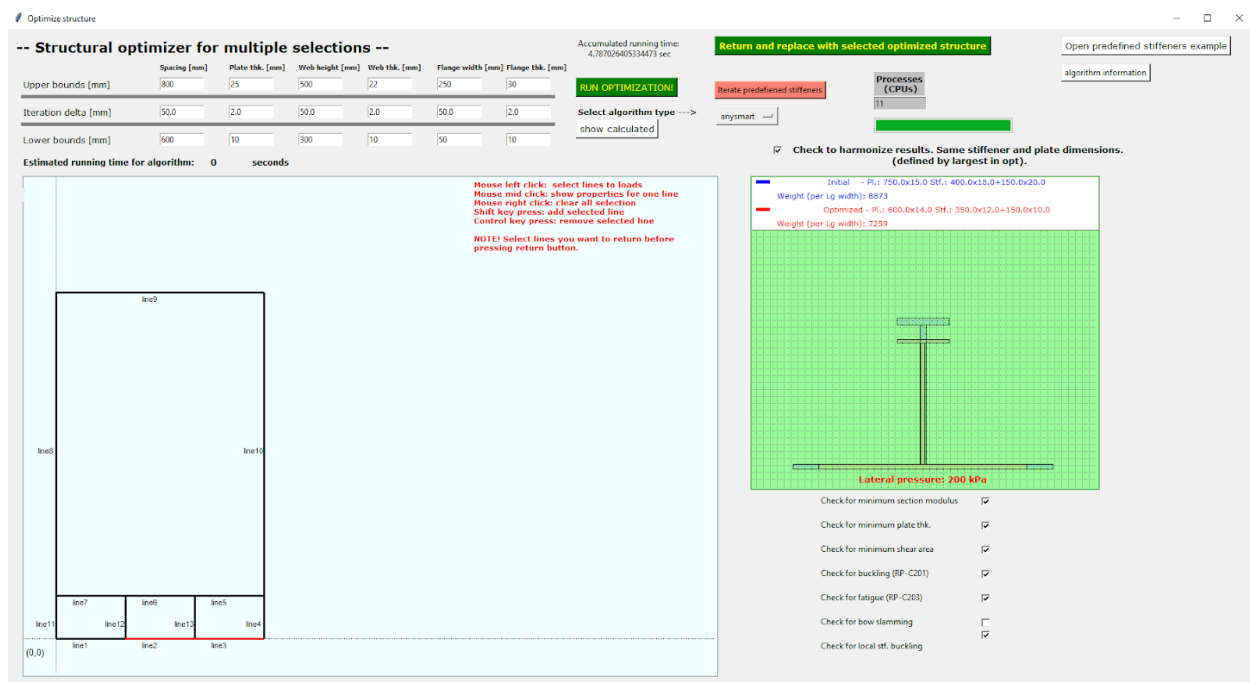
Check for minimum section modulus	<input checked="" type="checkbox"/>
Check for minimum plate thk.	<input checked="" type="checkbox"/>
Check for minimum shear area	<input checked="" type="checkbox"/>
Check for buckling (RP-C201)	<input checked="" type="checkbox"/>
Check for fatigue (RP-C203)	<input checked=""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

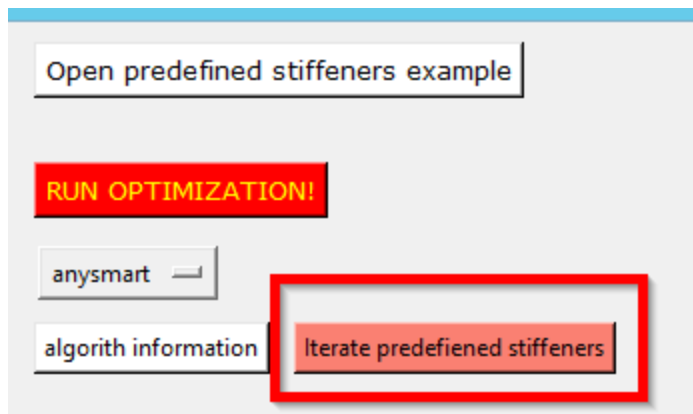
The optimization can be **harmonized**. That means that the largest dimension found in the multiple optimization is used for all selected. This is done after all

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).

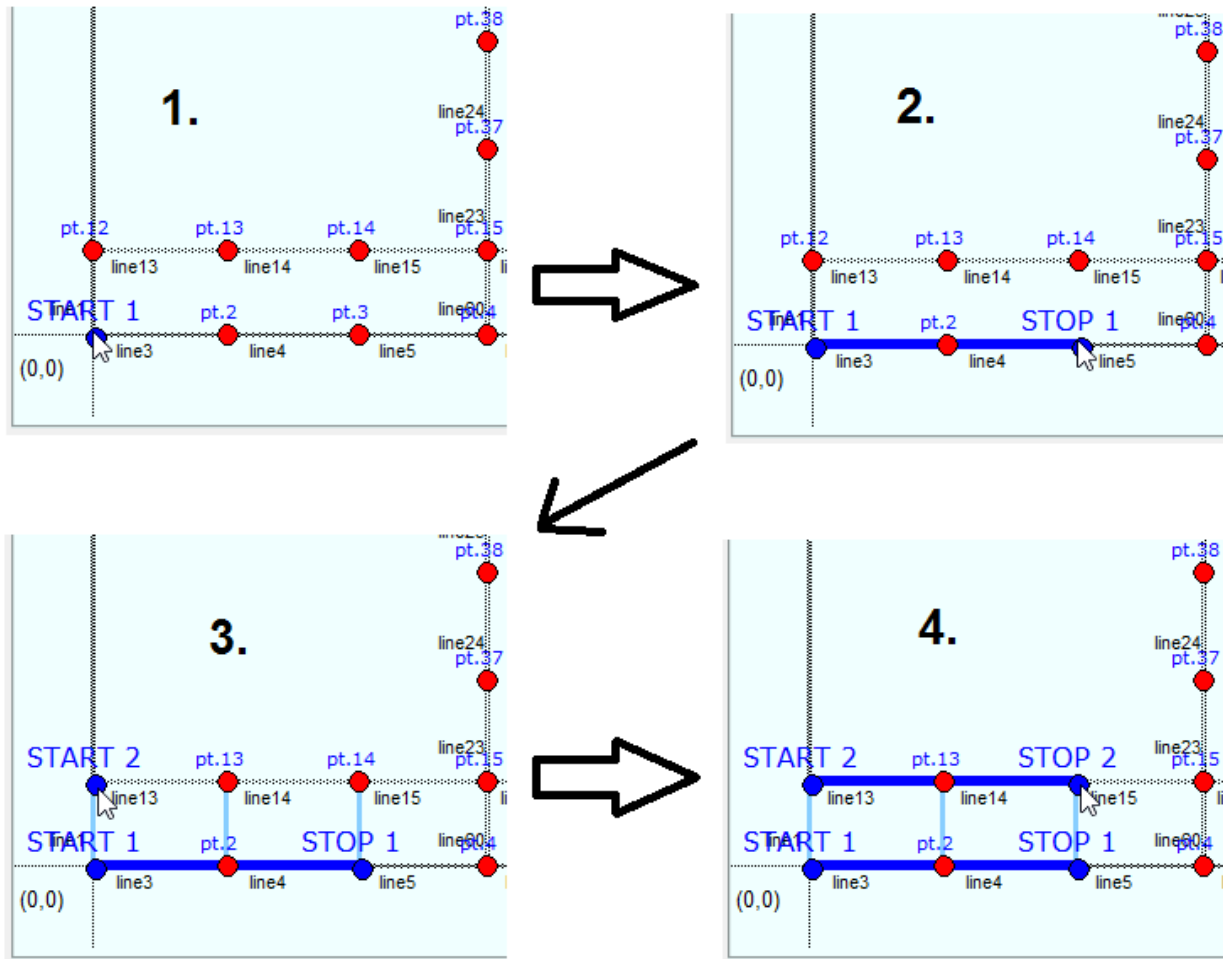
Span optimization

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



The optimization is started as follows.

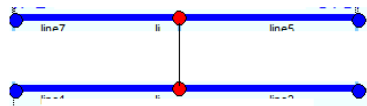
1. Start by clicking as illustrated next:



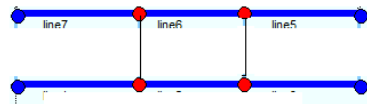
2. Then run optimization.

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.

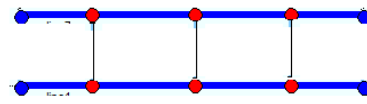
4 plate fields



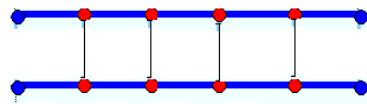
6 plate fields



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 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.

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="1.2"/>	<input type="text" value="0.8"/>
		Maximum span / Minimum span -> <input type="text" value="6"/> <input type="text" value="2"/>	

Results are presented as seen next.

RUN OPTIMIZATION!

anysmart

algorithm 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.

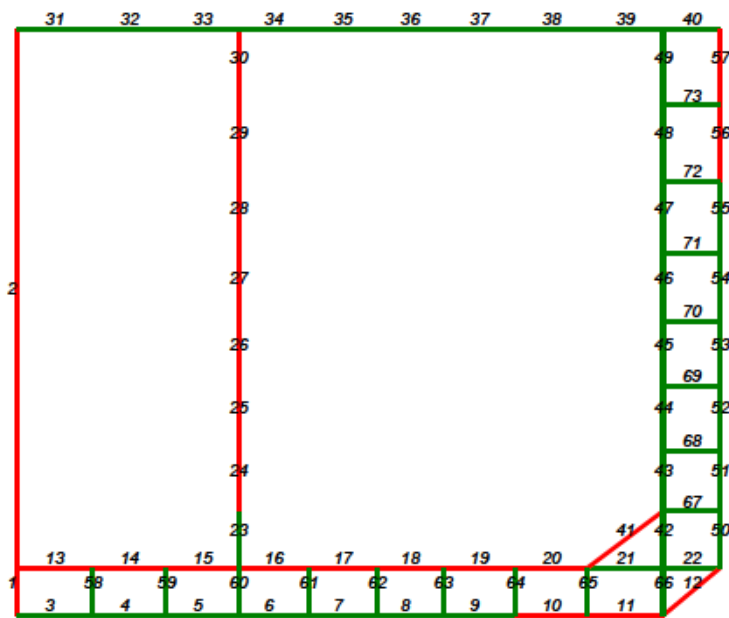
Reporting

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

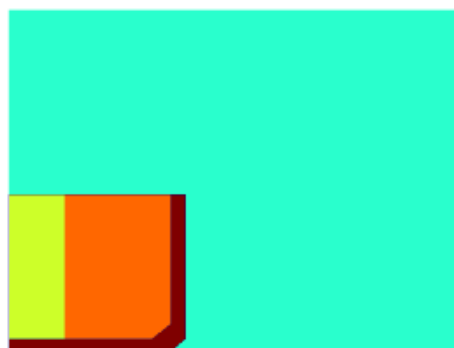
User: CEFANY

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

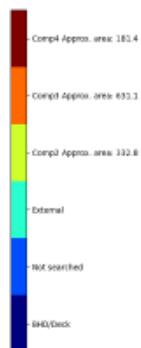


Compartments:

Compartments returned from search operation displayed below



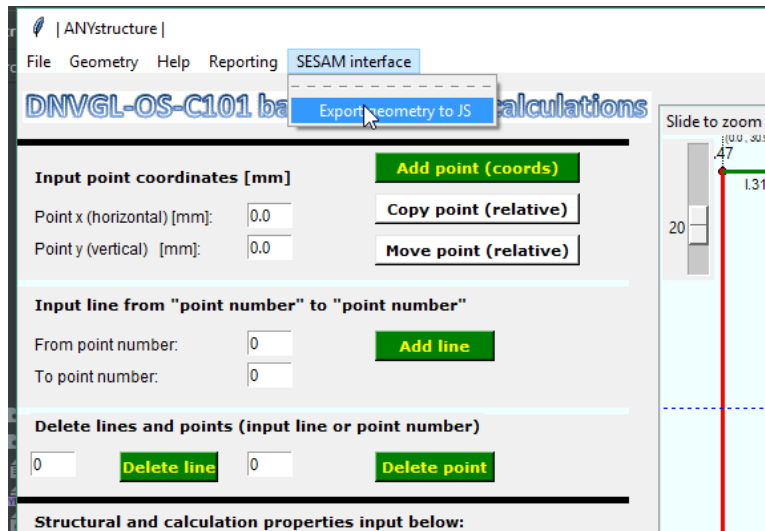
*Note calculations inaccuracies due to thickness of barriers (BHD/Deck)



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:

