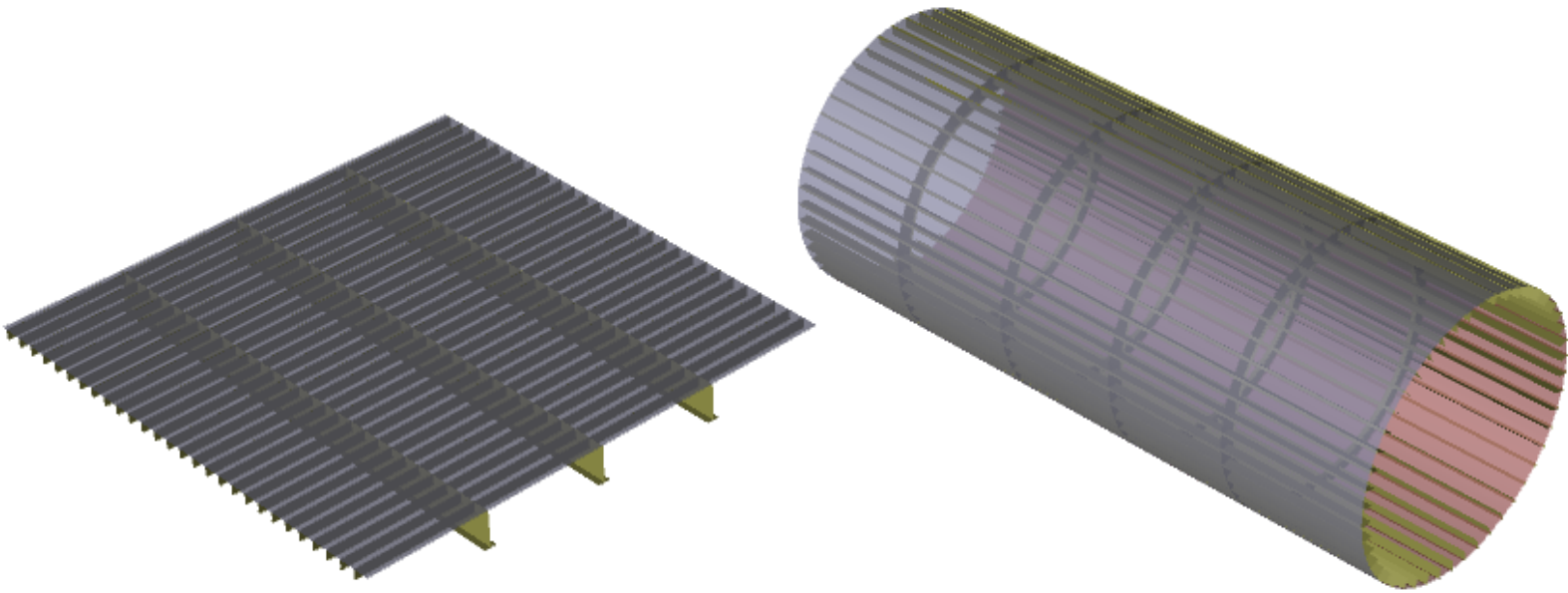




# ANYstructure



Documentation  
2022  
Version 3.4

	2
<b>Introduction</b>	<b>3</b>
<b>Theory</b>	<b>3</b>
<b>Modeling</b>	<b>4</b>
<b>Assigning properties</b>	<b>5</b>
Scale stresses	6
Cylinder input	8
Changing multiple properties	8
Display properties	9
<b>Define tanks</b>	<b>10</b>
COB and COG	12
<b>Setting accelerations</b>	<b>14</b>
<b>Define external pressures</b>	<b>15</b>
<b>Load combinations</b>	<b>16</b>
Changing load factors	17
<b>Buckling</b>	<b>18</b>
Prescriptive	18
PULS integration	19
ML buckling	21
Cylinder buckling	22
<b>Reviewing data</b>	<b>23</b>
Color coding	23
Shifting coordinates for visual purposes	24
Loads	24
Thickness and beam properties	26
Global stresses (buckling) and structure types	27
<b>Results</b>	<b>28</b>
<b>Optimization</b>	<b>29</b>
Optimization iteration by predefined stiffeners	29
Single optimization	30
Multiple optimization	33
Span optimization	34
<b>Reporting</b>	<b>39</b>
General	39
Table	40
<b>Export to JS</b>	<b>40</b>
<b>Changing the GUI</b>	<b>41</b>

# 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>
- DNV-RP-C202 - Buckling Strength of Shells
- PULS (Panel Ultimate Limit State)
- [DNVGL-CG-0128 Buckling](#)

# Modeling

Modeling is done in the Geometry tab.

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)

| ANYstructure |

File Geometry Reporting SESAM interface Help GUI

Geometry Line properties Properties tools Compartments and loads Information Help

No information on project provided. Input here.

---

**Input point coordinates [mm]**

Point x (horizontal) [mm]:

Point y (vertical) [mm]:

Show point names in GUI ☐

---

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

Line from point:

Line to point:

---

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

Line number (left click):

Point number (right click):

---

**Shift coordinate labeling [mm]:**

Used if you want a different origin of the reported coordinates.  
Does not affect loads.

y shift:

x shift:

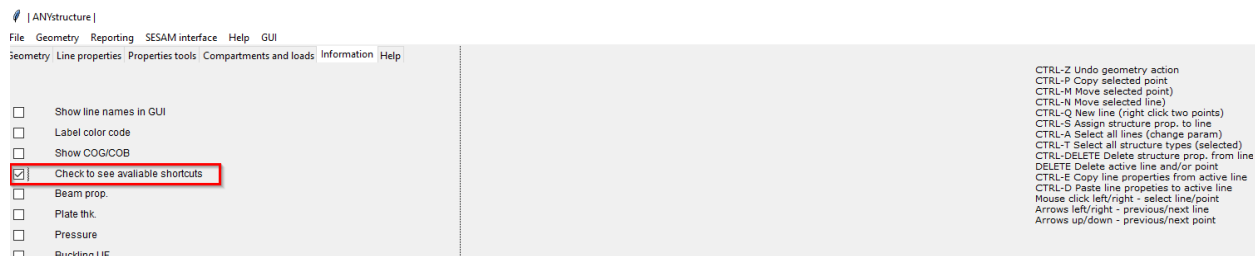
Use shifted coordinates ☐

Speed up your modeling significantly by using the shortcuts:

CTRL-Z	Undo modeling
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.



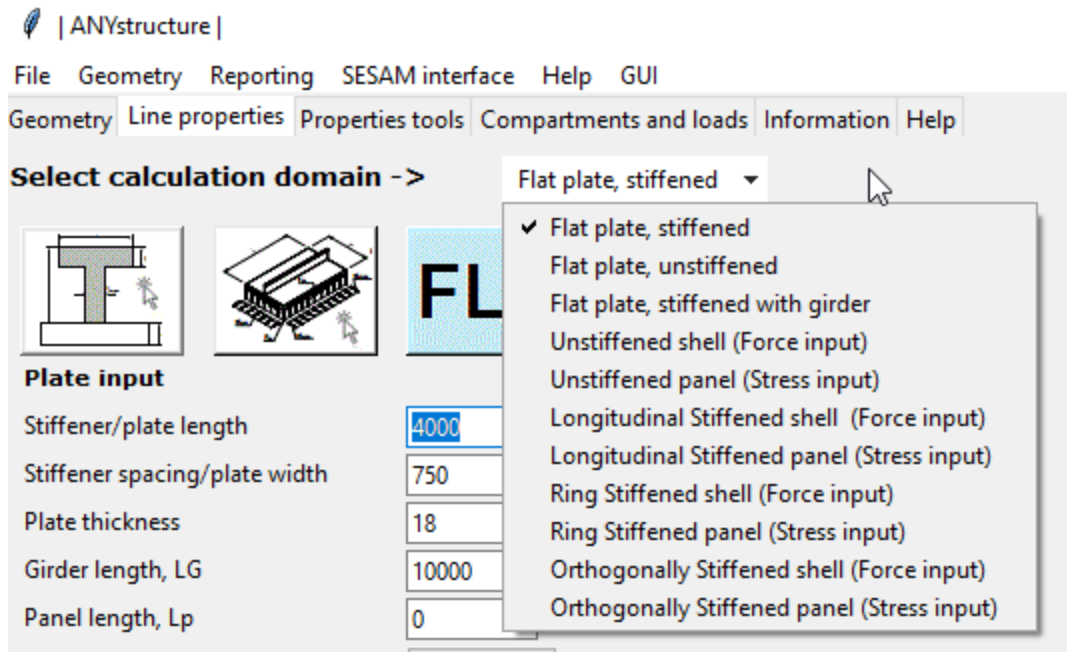
## Assigning properties

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.

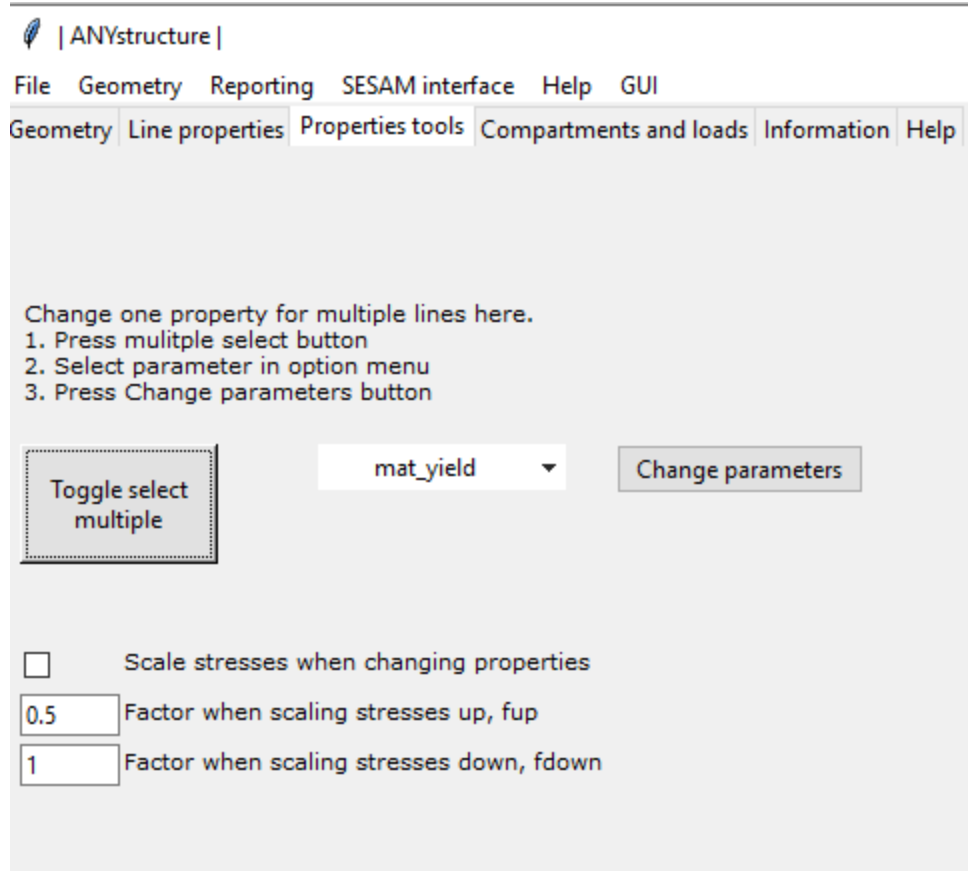
The first value to be set is the structure type. The selection is either a flat plate or some type of cylindrical structure (all others). The dropdown list is shown next. More details on the cylindrical structures.



**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”.**

## 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 ( $\sigma_X$ ), transversal stress ( $\sigma_Y$ ) and shear stress ( $\tau_{XY}$ ).

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

## Cylinder input

### Variables

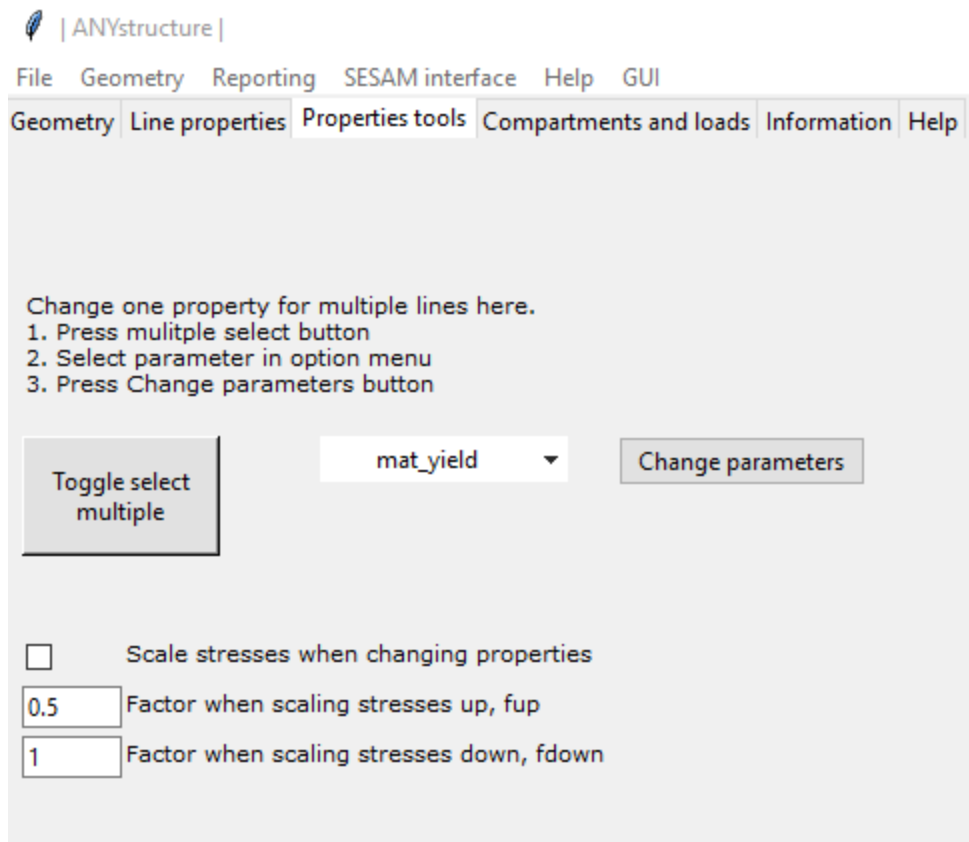
Thickness	t	Thickness of cylinder
Yield stress	fy	Material yield stress
Shell Radius (middle of plate)	r	Radius of cylinder
Distance between rings	l	Distance between ring stiffeners. If there are no ring stiffeners, the value is the same as cylinder length.
Length of shell	L	Length of the cylinder.
Total cylinder length	Lc	Used when the input is a complete cylinder (not panel). Total length of cylinder.
Eff. Buckling length factor	k	Effective length buckling factor.
Length between girders	Lh	Used when a heavy ring frame is applied. Distance between the girders/heavy ring frames.

## Changing multiple properties

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

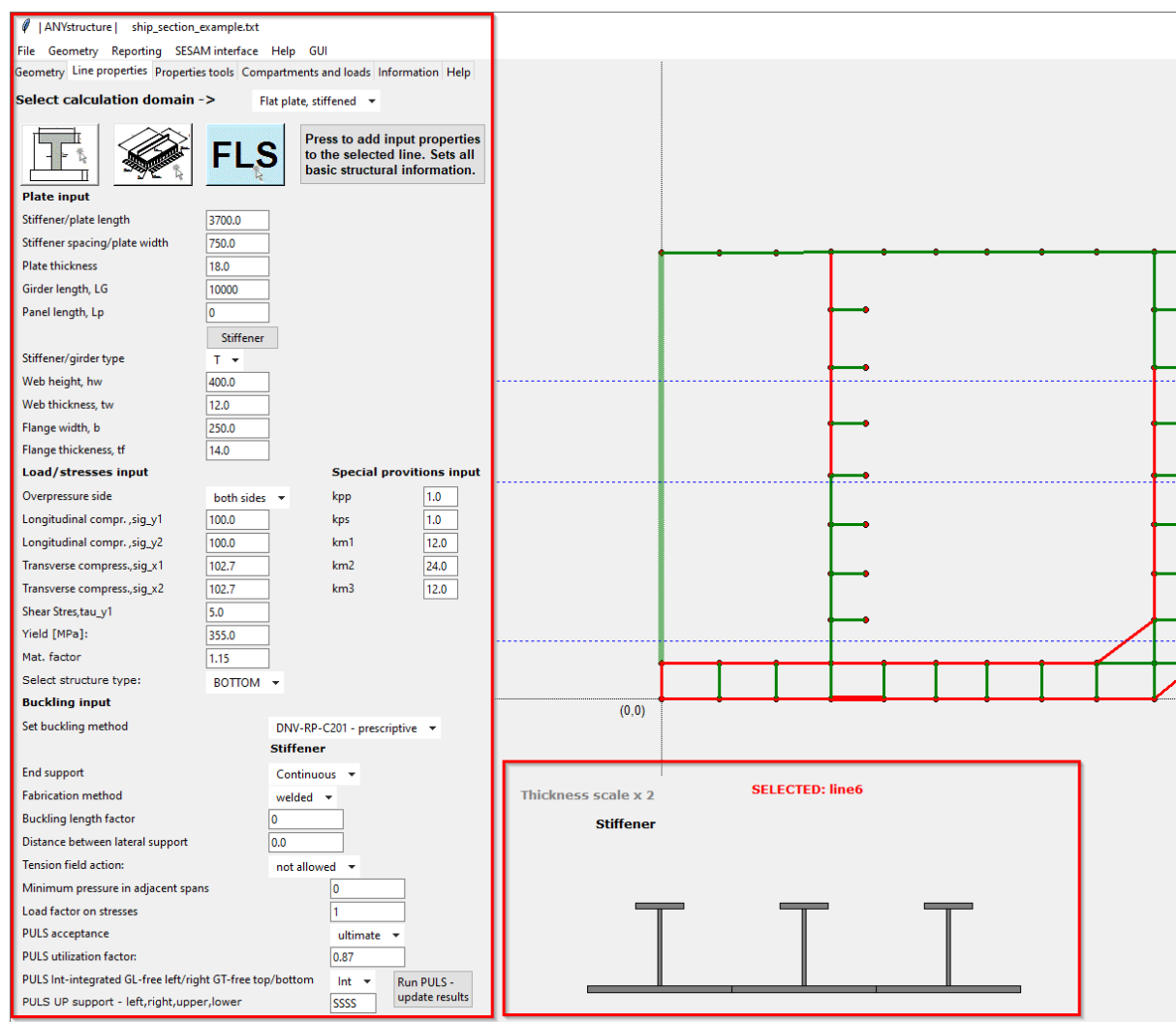
1. Press Toggle select multiple
2. Select the parameter to change
3. Select the lines to change. Click single lines, CTRL-A or CTRL-T (see shortcuts)
4. Press Change multi. param.





## Display properties

If you click a line properties are shown in the input fields under the “Line properties” tab. In addition a visual representation of the structure is shown.



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

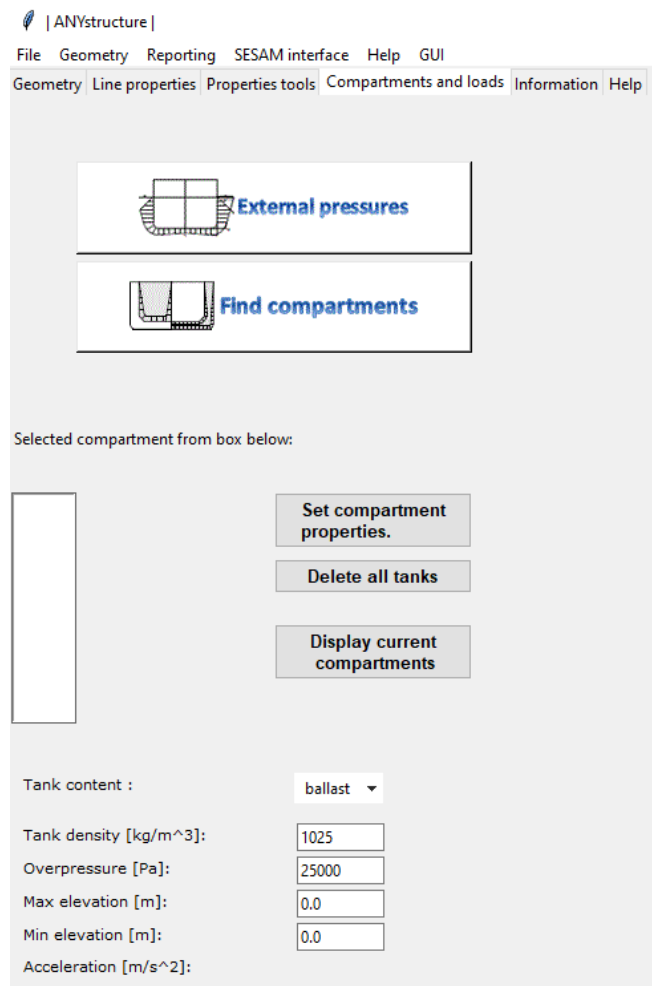
## Define tanks

Compartment and loads found in the “Compartments and loads” tab.

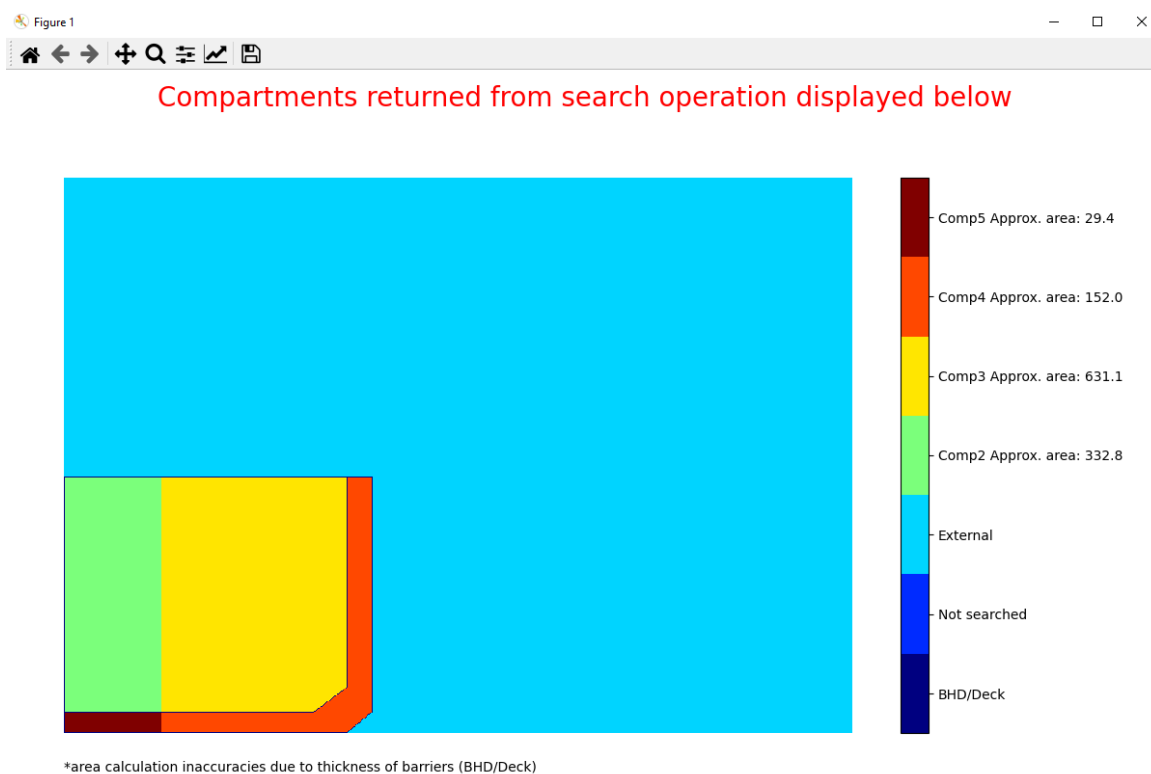
Tanks are searched for when clicking “Find compartments”. Non watertight structures are ignored. For information on structure types click “Show structure types”.

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

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

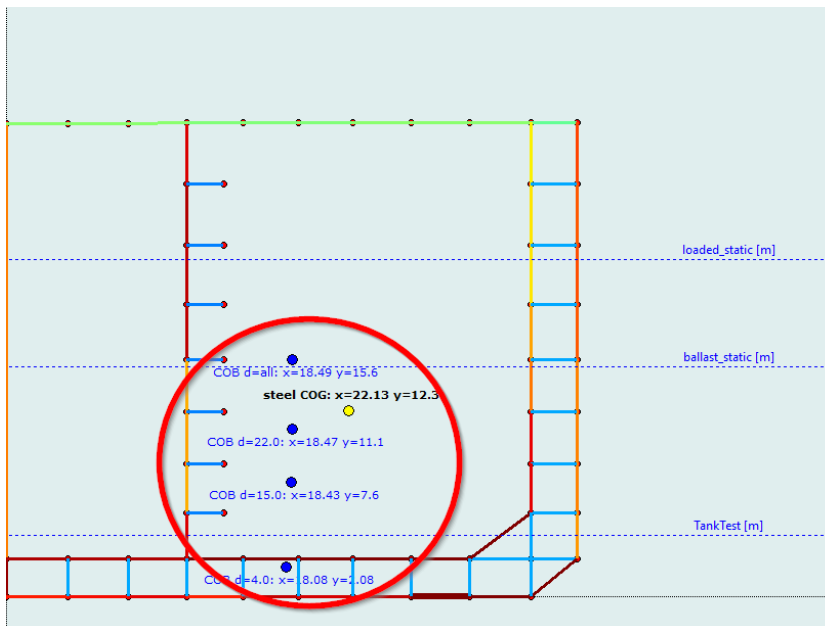
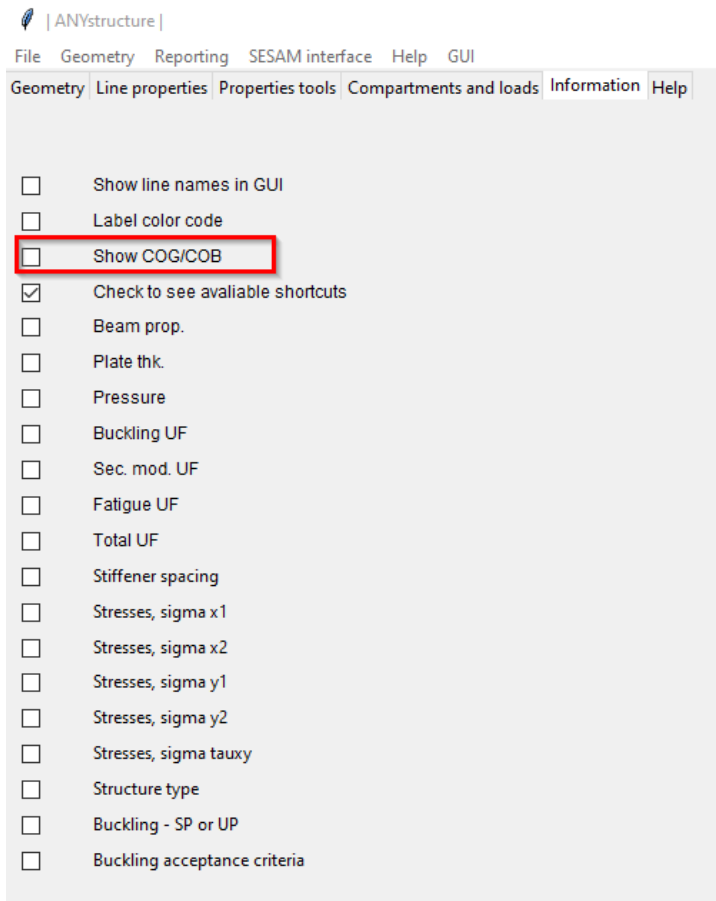


**If you press “Display current compartments” after doing a compartment search, the result of the search is illustrated as seen next. Approximate area of the respective compartments is also shown.**

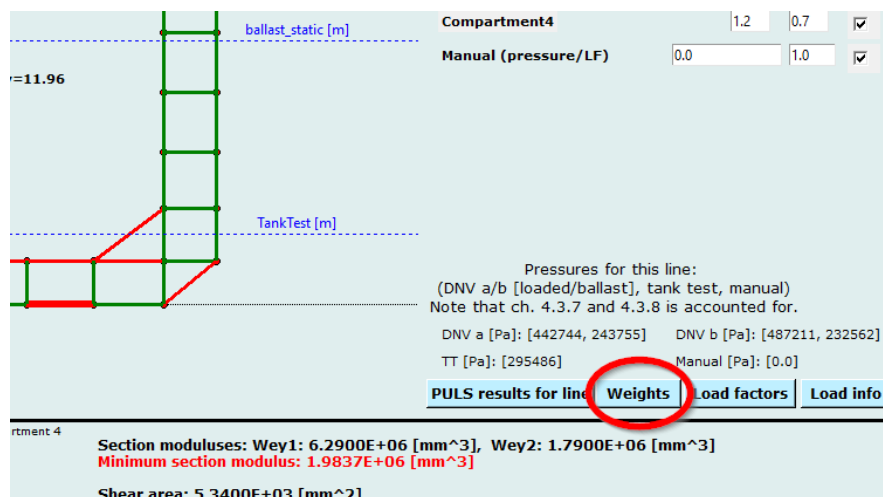


## COB and COG

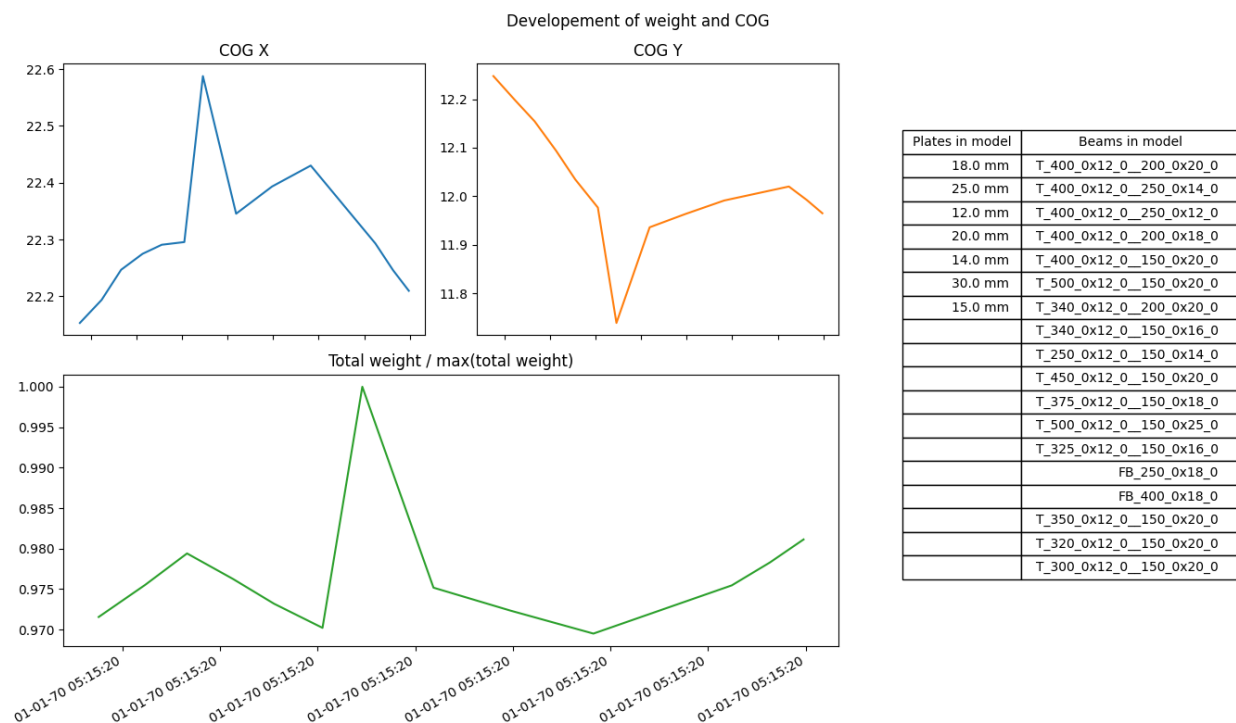
COG is calculated when structure properties are defined. COB for various drafts is calculated when tank search is completed.



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 apply to tank content. It is set in the upper right corner as seen next.

— □ ×

Static and dynamic accelerations **line10**

Static acceleration [m/s<sup>2</sup>]:

Dyn. acc. loaded [m/s<sup>2</sup>]:

Dyn. acc. ballast [m/s<sup>2</sup>]:

**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]:

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\_bbside\_top  
loaded\_side\_shell  
loaded\_above\_wl  
ballast\_bbside\_top  
ballast\_side\_shell  
ballast\_above\_wl  
static\_15m  
static\_8m\_tt  
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 is 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

Name:	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"

Condition a) - Static load factor well defined loads

Condition a) - Dynamic load factor

Condition b) - Static load factor "unknown loads"

Condition b) - Static load factor well defined loads

Condition b) - Dynamic load factor

Tank test) - Static load factor "unknown loads"

Tank test) - Static load factor well defined loads

Tank test) - Dynamic load factor

**Return specified load factors and change existing**

**Table 1 Load factors  $\gamma_f$  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  $\gamma_f$  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

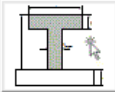


Buckling calculations as per DNV-RP-C203.

ANYstructure |

File Geometry Reporting SESAM interface Help GUI

Geometry Line properties Properties tools Compartments and loads Information Help

Select calculation domain -> Flat plate, stiffened

Press to add input properties to the selected line. Sets all basic structural information.

**Plate input**

Stiffener/plate length: 4000

Stiffener spacing/plate width: 750

Plate thickness: 18

Girder length, LG: 10000

Panel length, Lp: 0

Stiffener

Stiffener/girder type: T

Web height, hw: 400

Web thickness, tw: 12

Flange width, b: 150

Flange thickness, tf: 20

**Load/stresses input**

Overpressure side: both sides

Longitudinal compr., sig\_y1: 90

Longitudinal compr., sig\_y2: 90

Transverse compress., sig\_x1: 40

Transverse compress., sig\_x2: 40

Shear Stress, tau\_y1: 5

Yield [MPa]: 355

Mat. factor: 1.15

Select structure type: BOTTOM

**Special provisions input**

kpp: 1

kps: 1

km1: 12

km2: 24

km3: 12

**Buckling input**

Set buckling method: DNV-RP-C201 - prescriptive

End support: DNV-RP-C201 - prescriptive

Fabrication method: DNV PULS

Buckling length factor: 0.0

Distance between lateral support: 0.0

Tension field action: not allowed

Minimum pressure in adjacent spans: 0.0

Load factor on stresses: 1

PULS acceptance: buckling

PULS utilization factor: 0.87

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

PULS UP support - left,right,upper,lower: SSSS

Run PULS - update results

## 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 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. Note that the “PULS utilization factor” does not apply. The CL (classification) neural network will use  $1/1.15 = 0.87$  as material factor and get the acceptance from that. If you need other acceptance criterias, contact Audun ([audunarn@gmail.com](mailto:audunarn@gmail.com)). In that case the neural networks must be retrained for other acceptance levels.

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

ML buckling can be used for all optimization options.

## Cylinder buckling

Prescriptive buckling calculations according to DNV-RP-C202. The various calculated buckling modes are illustrated next.

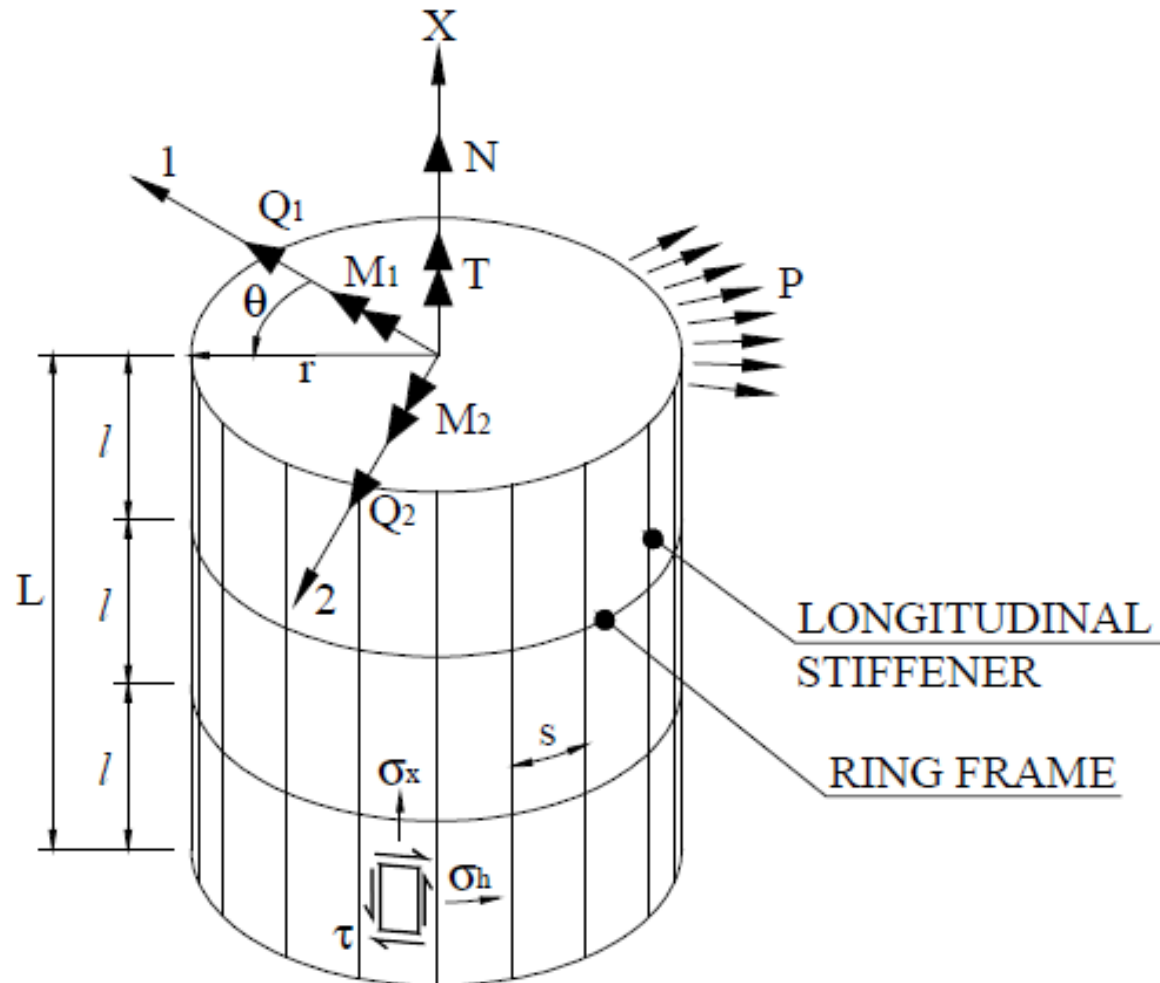


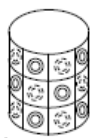


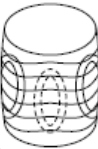
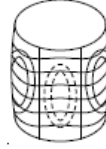




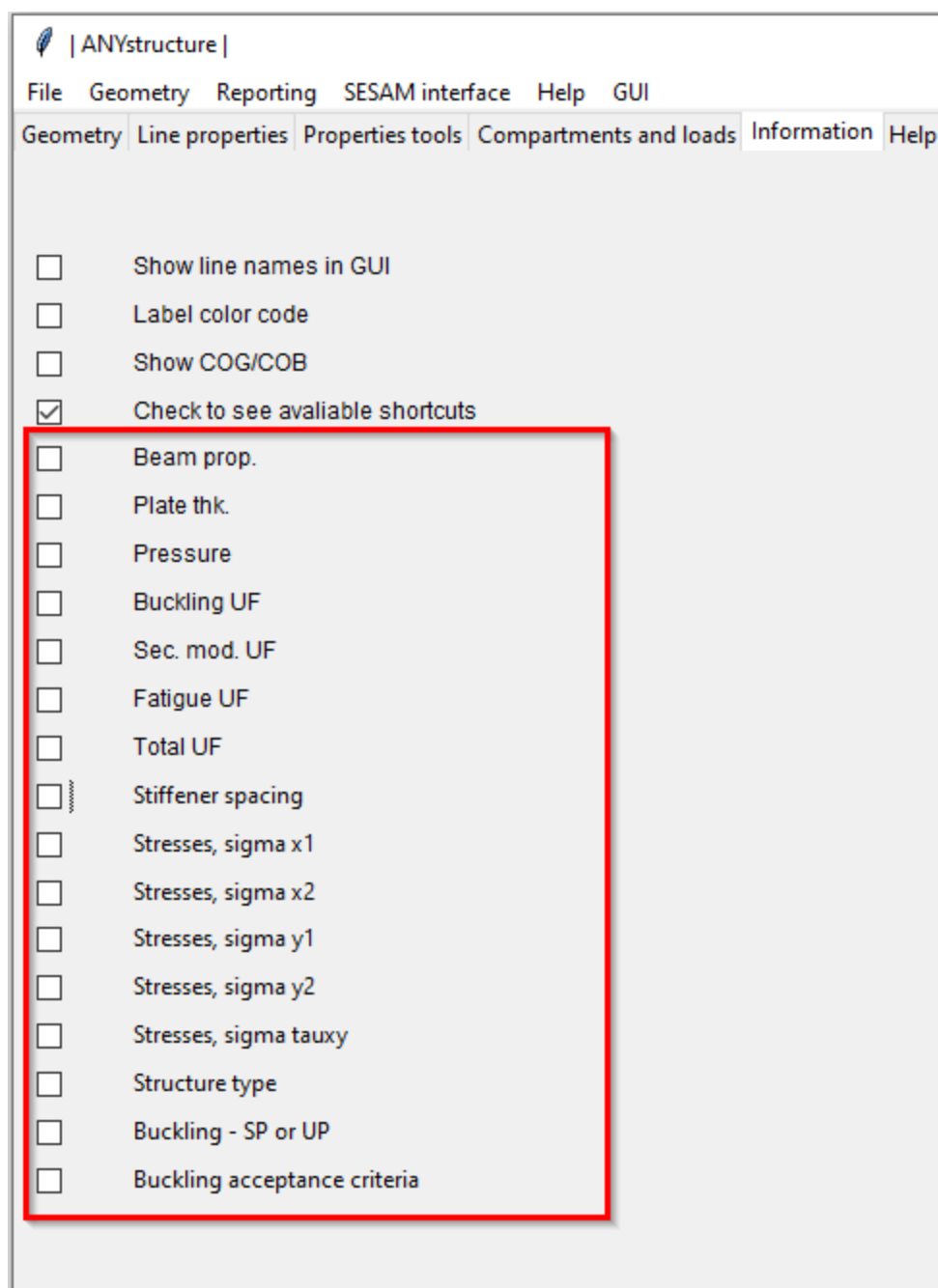


Table 1-1 Buckling modes for different types of cylinders			
Buckling mode	Type of structure geometry		
	Ring stiffened (unstiffened circular)	Longitudinal stiffened	Orthogonally stiffened
a) Shell buckling	 Section 3.4	 Section 3.3	 Section 3.3
b) Panel stiffener buckling		 Section 3.6	 Section 3.7
c) Panel ring buckling	 Section 3.5		 Section 3.7
d) General buckling			 Section 3.7
e) Column buckling	 Section 3.8	 Section 3.8	 Section 3.8

## Reviewing data

### Color coding

All color coding options are indicated next.



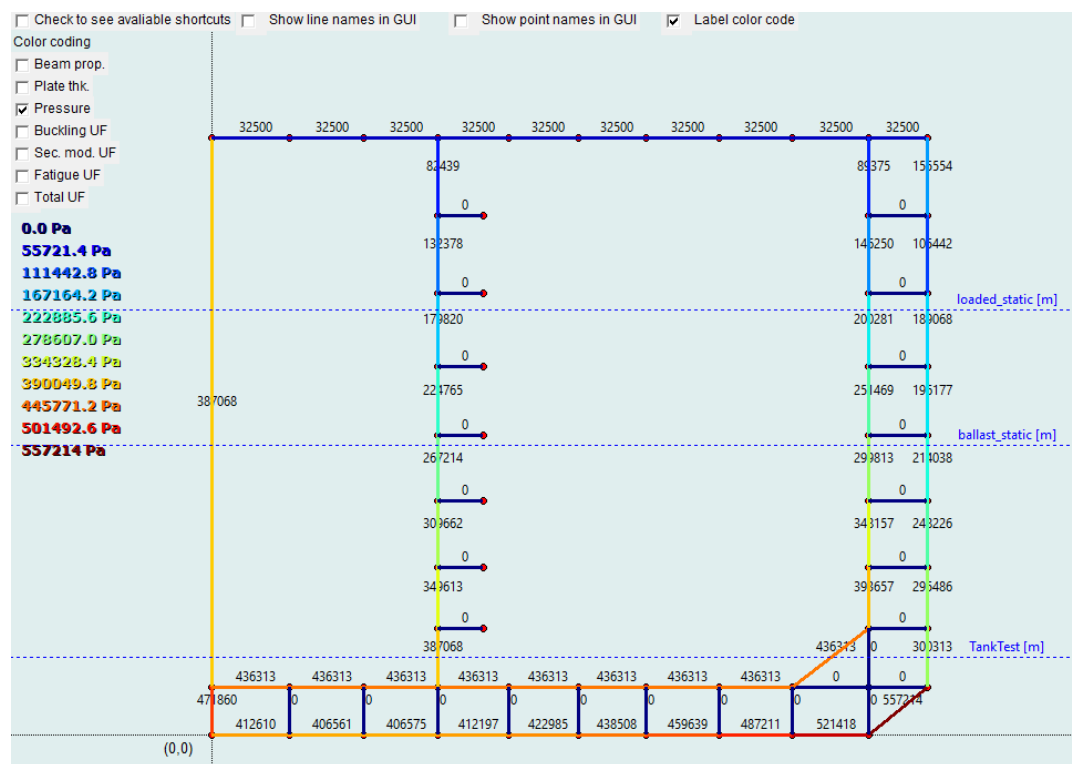
## Shifting coordinates for visual purposes

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 the 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.

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.





**Special provisions - DNV-OS-C101 - checks for section, web thickness and plate thickness.**

	Minimum value	Actual value	Accepted?
Section modulus check	<b>8.1018E+05 [mm<sup>3</sup>]</b>	<b>1.7400E+06 [mm<sup>3</sup>]</b>	<b>Ok</b>
Shear area check	<b>1.7920E+03 [mm<sup>2</sup>]</b>	<b>4.5360E+03 [mm<sup>2</sup>]</b>	<b>Ok</b>
Plate thickness check	<b>11.0 [mm]</b>	<b>18.0 [mm]</b>	<b>Ok</b>

**Buckling results DNV-RP-C201 - prescriptive - (plate, stiffener, girder):**

	Plate	Stiffener	Girder
Overpressure plate side	<b>0.352</b>	<b>0.641</b>	<b>0</b>
Overpressure stiffener side		<b>0.554</b>	<b>0</b>
Resistance between stiffeners		<b>0.865</b>	<b>0</b>
Shear capacity		<b>0.411</b>	
Maximum web height [mm]		410.0	0
Maximum flange width [mm]		228.0	0

**Fatigue results (DNVGL-RP-C203):****Total damage: NO RESULTS**

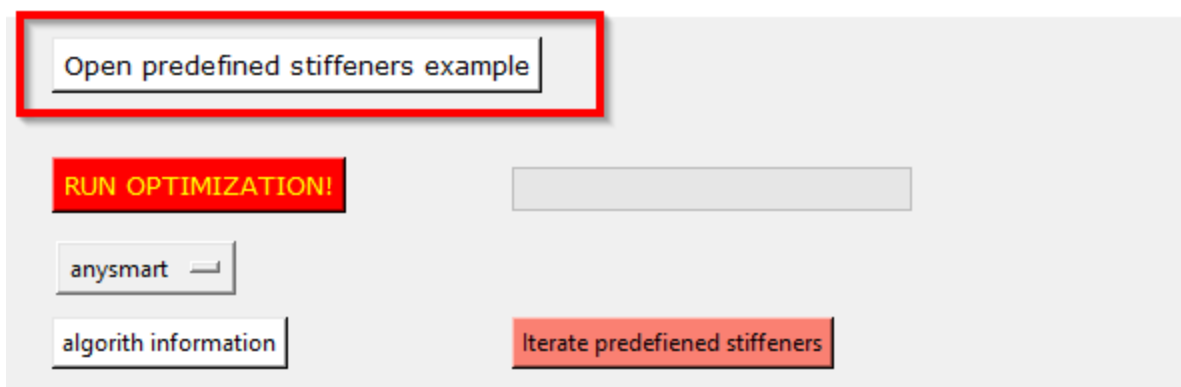
A combined utilization can be reviewed using color coding.

## 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. An 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 search. 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:

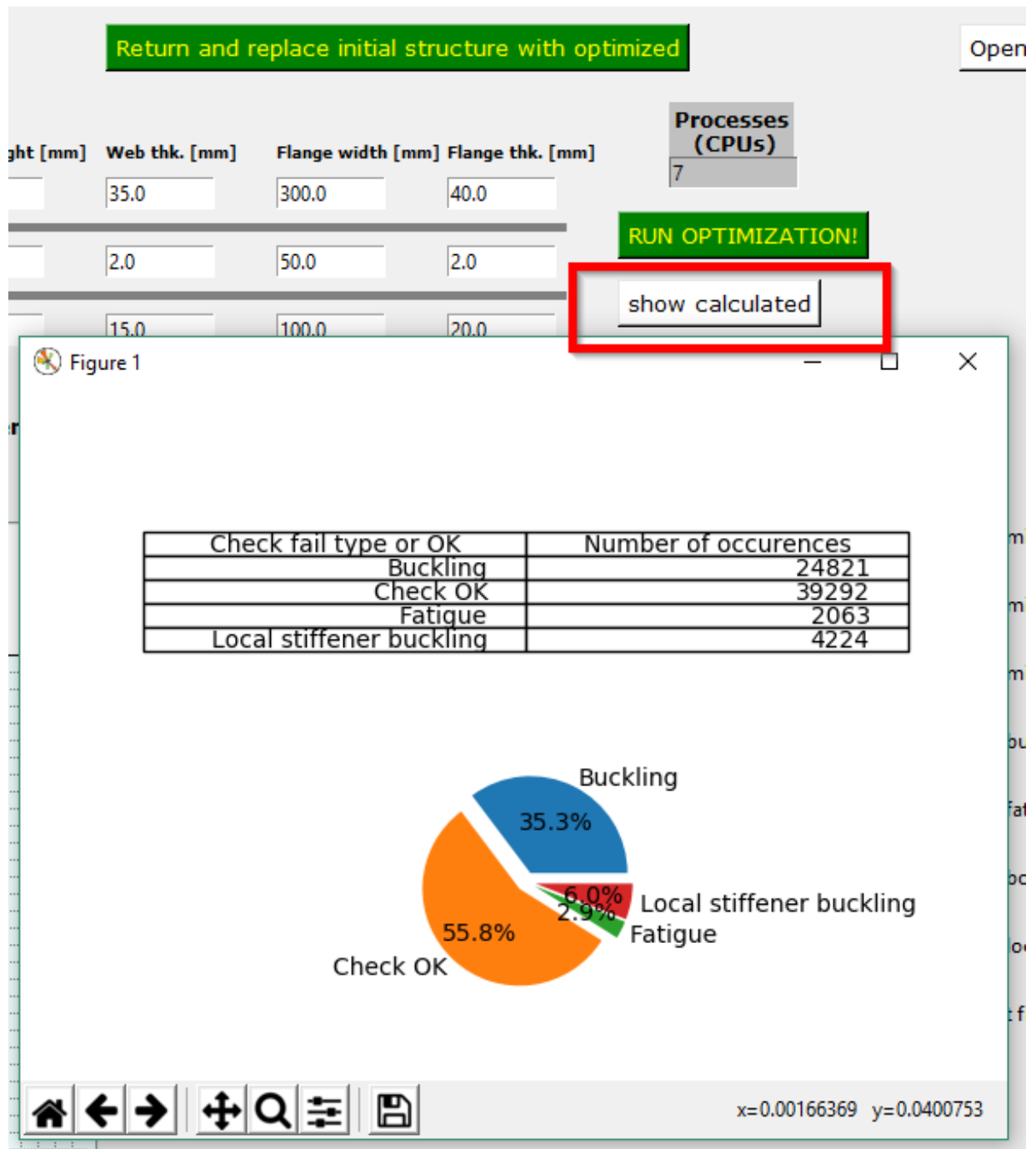
You can select the checks to be performed. PULS buckling can be used in optimization.

Remember to check the running time.

The weight filter ensures that only sections with a lower weight than the current minimum weight. This significantly speeds 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"/>

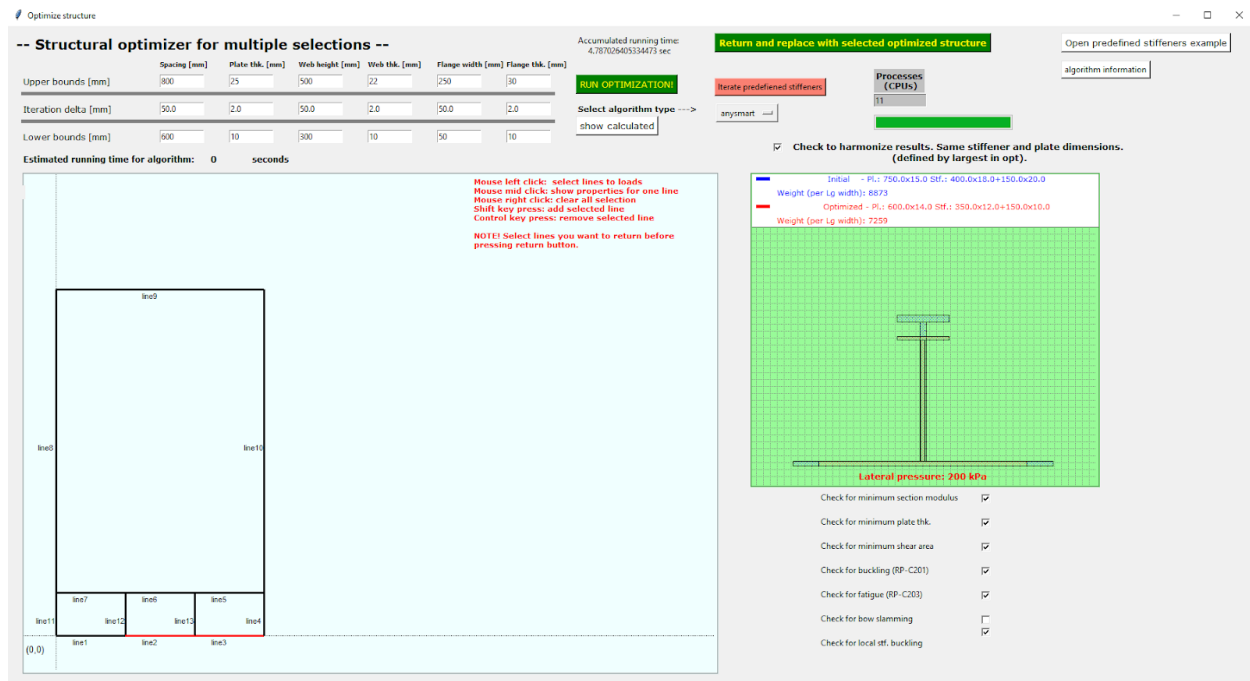
If you press the “show calculated” button, you will get an overview of how many are ok and how many failed (and what criteria first failed). One “occurrence” is a one checked plate/stiffener combination.



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 are 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 are 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.

Open predefined stiffeners example

**RUN OPTIMIZATION!**

anysmart

algorithm information

Iterate predefined stiffeners

Select number of panels: None

Select panel to plot: None

show calculated

Show previous results

Optimize structure

-- Plate field span optimizer for plate fields separated by frames. --

	Spacing [mm]	Plate thk. [mm]	Web height [mm]	Web thk. [mm]	Flange width [mm]	Flange thk. [mm]
Upper bounds [mm]	800	25	500	22	250	30
Iteration delta [mm]	50.0	2.0	50.0	2.0	50.0	2.0
Lower bounds [mm]	600	10	300	10	50	10

Estimated running time for algorithm not calculated.

☐ Harmonize stiffer spacing for section.

Processes (CPUs): 11

How to:  
For a double bottom structure:  
Click start point 1 -> click on point 1 (for example bottom plate)  
Click start point 2 -> click on point 2 (for example inner bottom)  
Run optimization! Wait for the results..... wait.... wait....

START 2 pt. 7 pt. 6 STOP 2

START 1 pt. 2 pt. 3 STOP 1

(0,0) line1 line2 line3

line5 line6 line7 line8 line9 line10

Open predefined stiffeners example

**RUN OPTIMIZATION!**

anysmart

algorithm information

Iterate predefined stiffeners

Select number of panels: None

Select panel to plot: None

show calculated

Show previous results

Check for minimum section modulus ☒

Check for minimum plate thk. ☒

Check for minimum shear area ☒

Check for buckling (RP-C201) ☒

Check for fatigue (RP-C203) ☒

Check for bow slamming ☒

Check for local stf. buckling ☒

Factor when scaling stresses up, fup 0.5

Factor when scaling stresses up, fdown 1

Frame (girder data) for weight calculation:

Girder thickness 0.010

Stiffener height 0.25

Stiffener thickness 0.015

Stf. flange width 0

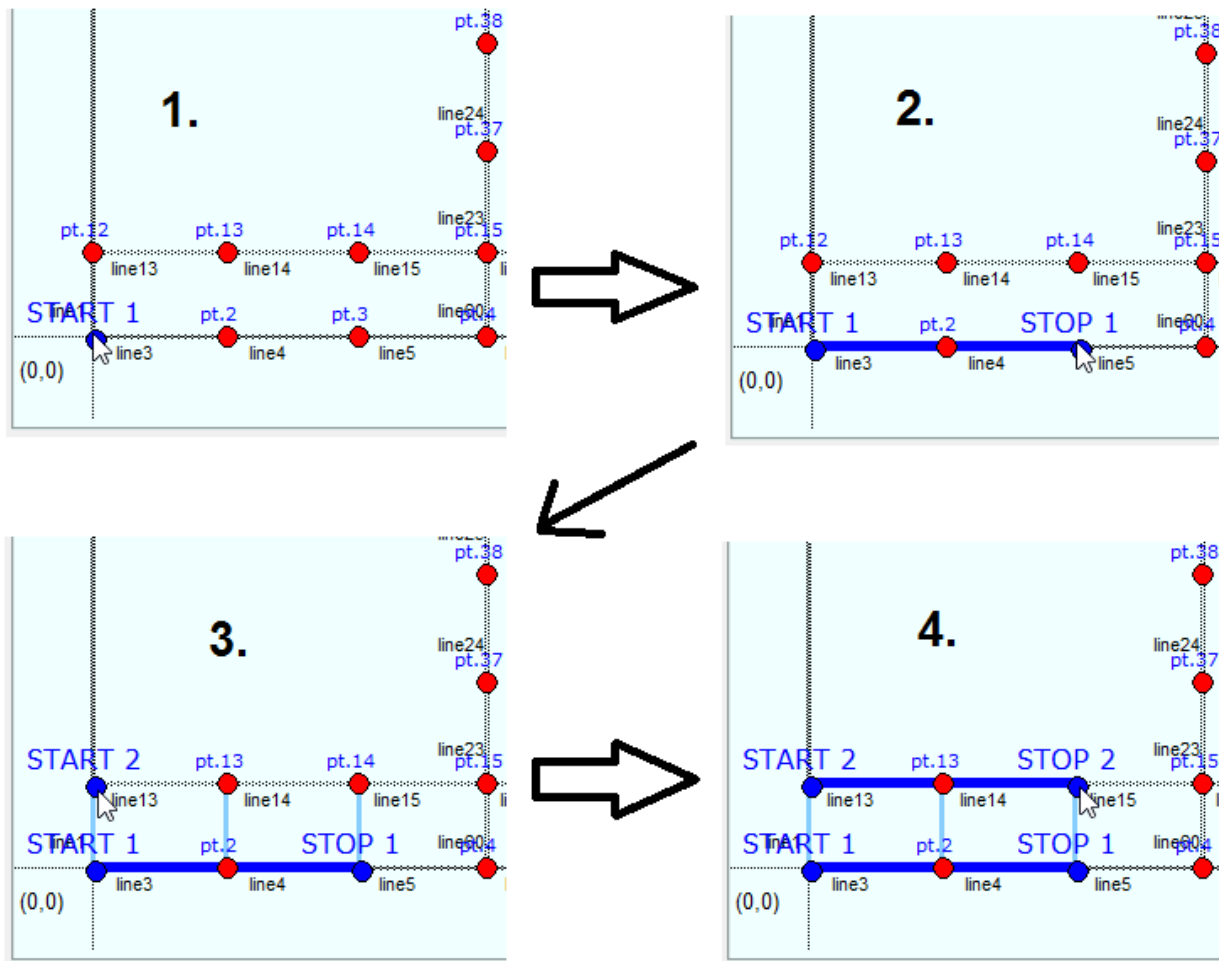
Stf. flange thickness 0

For weight calculation of girder: Max span mult / Min span mult 1.1 0.9

Maximum span / Minimum span -> 0 2

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.

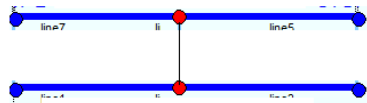
Check for minimum section modulus	<input checked="" type="checkbox"/>	<b>Frame (girder data) for weight calculation:</b>	
Check for minimum plate thk.	<input checked="" type="checkbox"/>	Girder thickness	<input type="text" value="0.018"/>
Check for minimum shear area	<input checked="" type="checkbox"/>	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 thickness	<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	
Factor when scaling stresses up, fup	<input type="text" value="0.5"/>	<input type="text" value="1.1"/>	<input type="text" value="0.9"/>
Factor when scaling stresses up, fdwn	<input type="text" value="1"/>	Maximum span / Minimum span ->	<input type="text" value="6"/> <input type="text" value="2"/>

3. Start the calculation

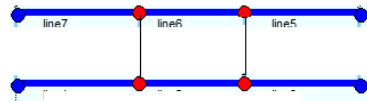
The program will calculate variations of even spans in your structure as illustrated next.

This is an example and the number of plate fields may vary.

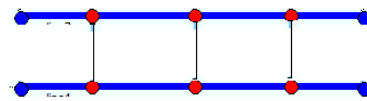
#### 4 plate fields



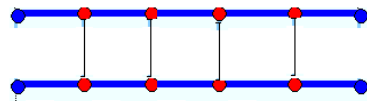
#### 6 plate fields



#### 8 plate fields



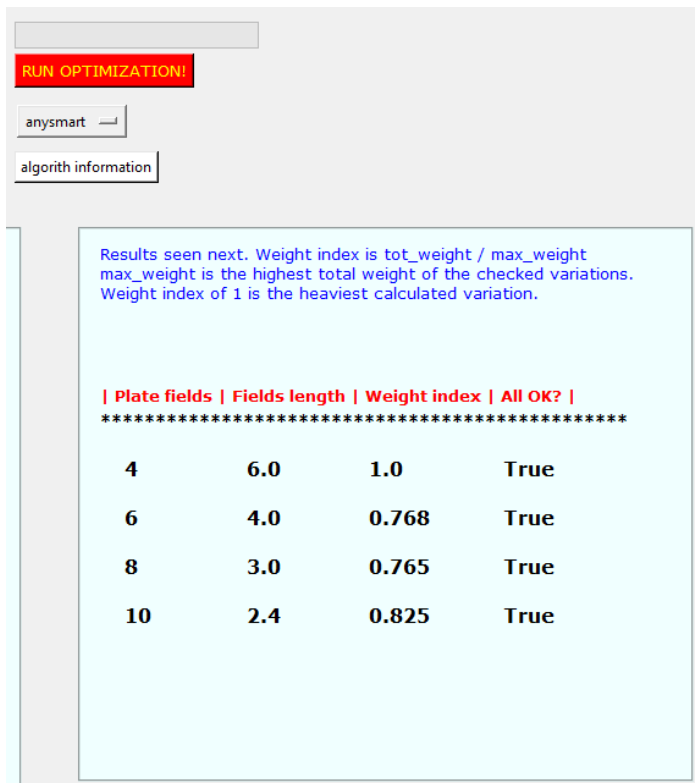
#### 10 plate fields



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.

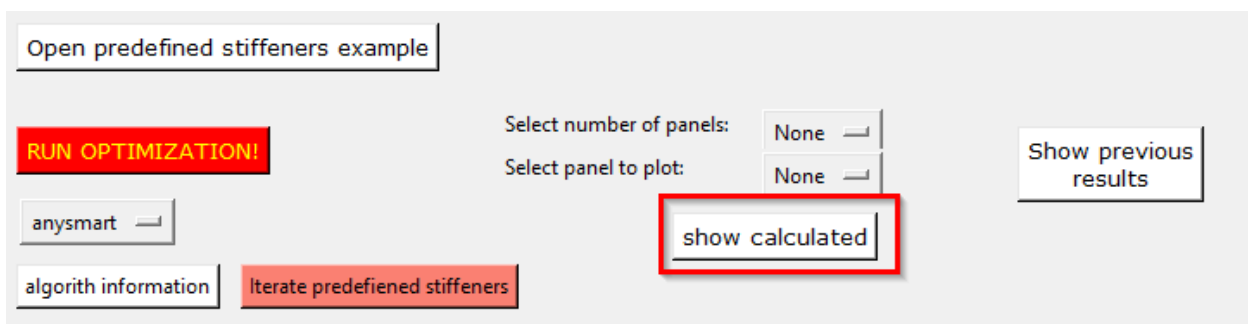
Minimum span and maximum span is the minimum and maximum span of the plate fields in meters.

Results are presented as seen next.



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 the 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.

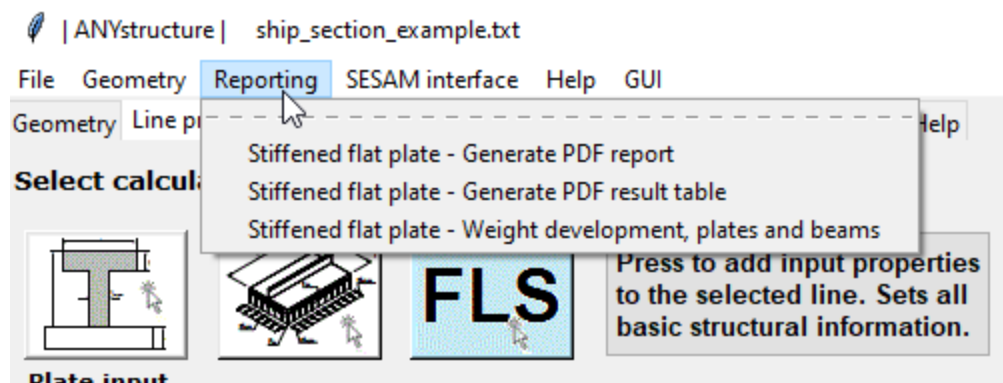


Now close the window. Results are not currently returned to the main window.

Detailed results, printed after running, looks like this :

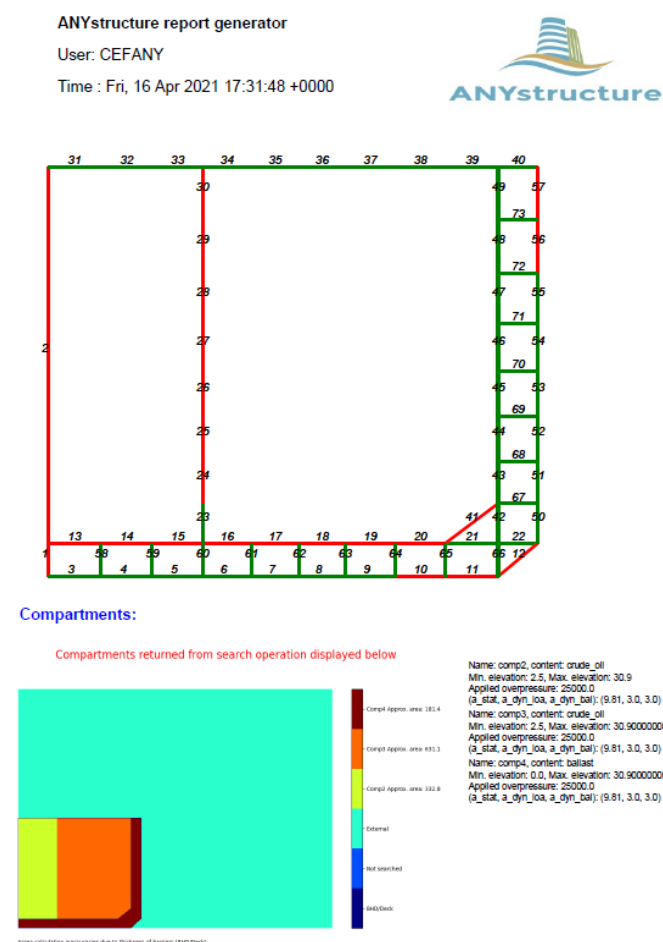
[illegible]

# 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.



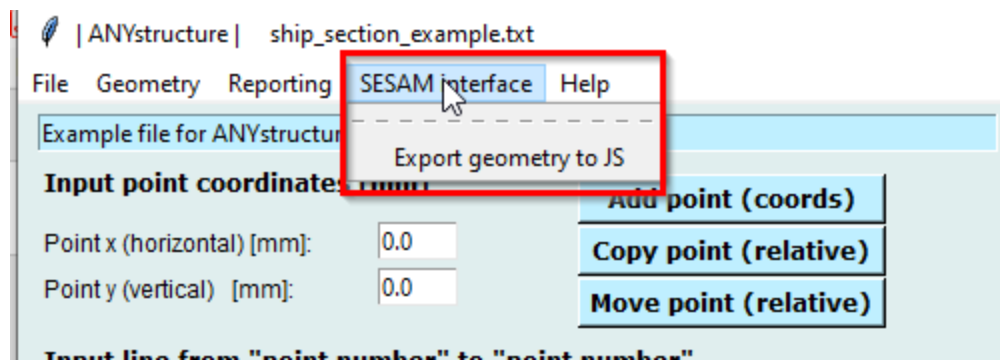
## Table

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

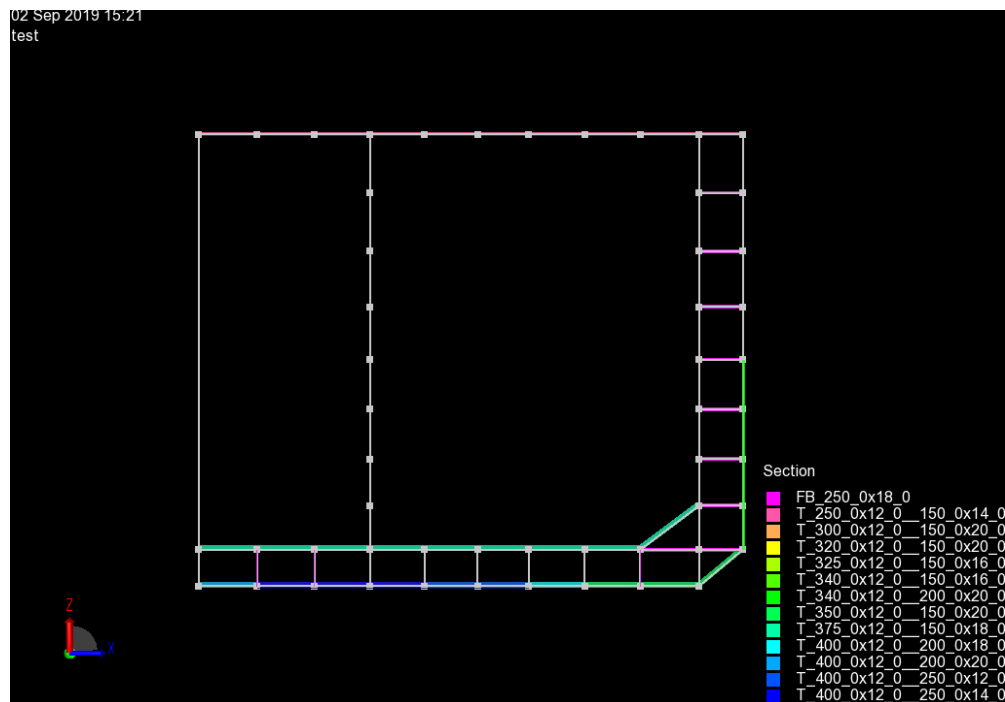
## Export to JS

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:





## Changing the GUI

Various GUI modifications can be selected as seen next.

