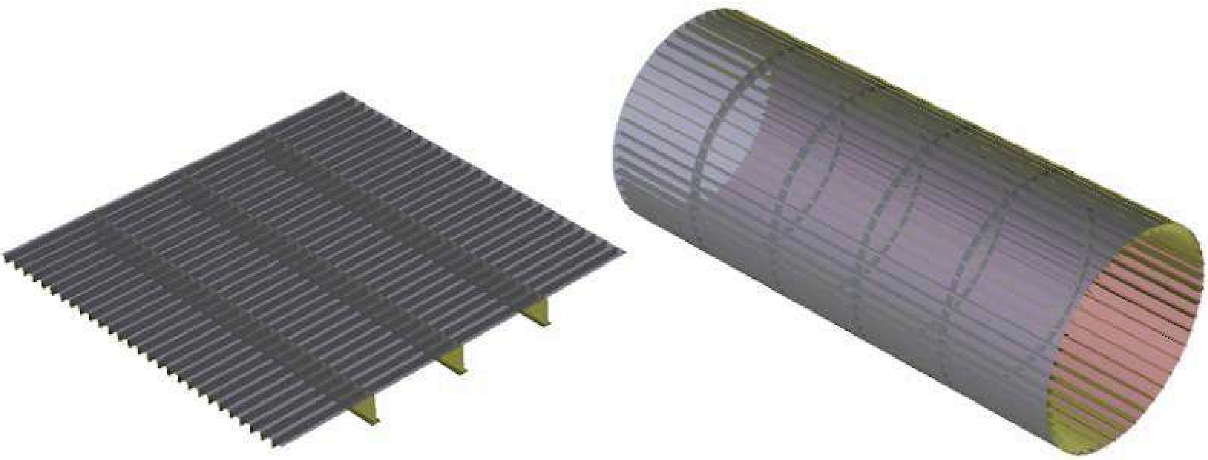
ANYstructure



Documentation

2026

Version 6

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)

The AIP is documented at readthedocs:

https://anystructure.readthedocs.io/en/latest/

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

ANYstructure API

See:

https://anystructure.readthedocs.io/en/latest/

From ANYstructure 5.

Basic calculations can also be performed using the ANYstructure API. The various input is

```
from anystruct.api import CylStru, FlatStru


def flatten_numbers(value):
    if isinstance(value, dict):
        for child in value.values():
            yield from flatten_numbers(child)
    elif isinstance(value, (list, tuple)):
        for child in value:
            yield from flatten_numbers(child)
    elif value is not None and not isinstance(value, bool):
        try:
            yield float(value)
        except (TypeError, ValueError):
            return


def max_uf(results):
    values = list(flatten_numbers(results))
    return max(values) if values else None


def print_flat_report(flat):
    print("FLAT PLATE")
    print("Available buckling methods:", ",
".join(flat.get_available_buckling_methods()))

    flat.set_buckling_parameters(
        calculation_method="DNV-RP-C201 - prescriptive",
        buckling_acceptance="ultimate",
        pressure_side="both sides",
        fabrication_method_stiffener="Fabricated",
        fabrication_method_girder="Cold formed",
    )
    dnv = flat.get_buckling_results()
    print(f"DNV-RP-C201 max UF: {max_uf(dnv):.3f}")

    flat.set_buckling_parameters(
        calculation_method="SemiAnalytical S3/U3",
```

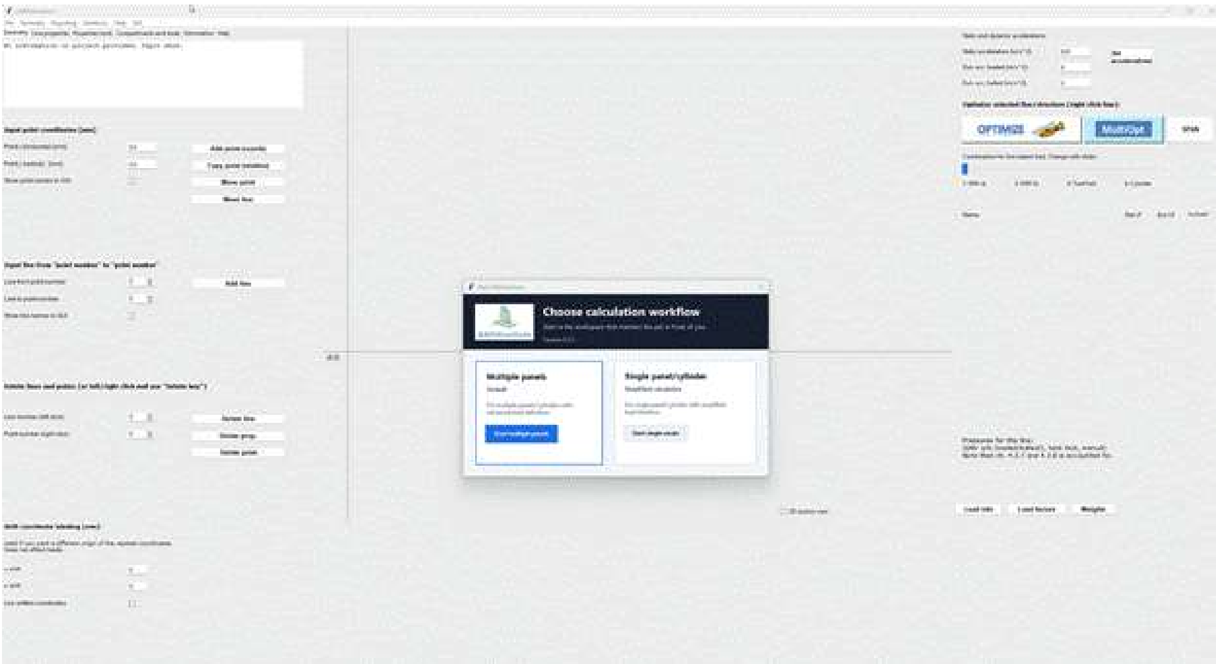
```
        buckling_acceptance="buckling",
    )
    semi = flat.get_buckling_results()
    print(f"SemiAnalytical selected UF: {semi['selected UF']:.3f}")
    print(f"SemiAnalytical valid: {semi['available']} ({semi['valid label']})")

    special = flat.get_special_provisions_results()
    for name, check in special.items():
        print(f"{name}: actual={check['actual']:.1f},
minimum={check['minimum']:.1f}")

    print("ML-Numeric: call set_ml_buckling_model(ml_algo) before using the ML
method.")


def print_cylinder_report(cylinder):
    results = cylinder.get_buckling_results()
    print("\nCYLINDER")
    for key in (
        "Unstiffened shell",
        "Longitudinal stiffened shell",
        "Ring stiffened shell",
        "Heavy ring frame",
        "Column stability UF",
    ):
        value = results.get(key)
        if value is not None:
            print(f"{key}: {float(value):.3f}")
    print("Stiffener check:", results.get("Stiffener check"))


flat = FlatStru("Flat plate, stiffened with girder")
flat.set_material(mat_yield=355, emodule=210000, material_factor=1.15,
poisson=0.3)
flat.set_plate_geometry(spacing=700, thickness=18, span=4000)
flat.set_stresses(
    pressure=0.2,
    sigma_x1=50,
    sigma_x2=50,
    sigma_y1=100,
    sigma_y2=100,
    tau_xy=5,
)
flat.set_stiffener(hw=360, tw=12, bf=150, tf=20, stf_type="T", spacing=700)
flat.set_girder(hw=600, tw=15, bf=220, tf=25, stf_type="T", spacing=2800)
flat.set_fixation_parameters(kpp=1, kps=1, km1=12, km2=24, km3=12)
flat.set_puls_parameters(
    sp_or_up="SP",
    puls_boundary="Int",
    stiffener_end="Continuous",
    up_boundary="SSSS",
)

cylinder = CylStru("Orthogonally Stiffened shell")
```

```
cylinder.set_material(mat_yield=355, emodule=210000, material_factor=1.15,
poisson=0.3)
cylinder.set_imperfection(delta_0=0.005)
cylinder.set_fabrication_method(stiffener="Fabricated", girder="Fabricated")
cylinder.set_end_cap_pressure_included_in_stress(is_included=True)
cylinder.set_uls_or_als(kind="ULS")
cylinder.set_shell_geometry(
   radius=6500,
   thickness=24,
   distance_between_rings=3300,
   tot_length_of_shell=20000,
)
cylinder.set_shell_buckling_parmeters(eff_buckling_length_factor=1.0)
cylinder.set_length_between_girder(val=3300)
cylinder.set_panel_spacing(val=680)
cylinder.set_longitudinal_stiffener(hw=260, tw=23, bf=49, tf=28,
stf_type="L-bulb", spacing=680)
cylinder.set_ring_girder(hw=500, tw=15, bf=200, tf=25, stf_type="T",
spacing=3300)
cylinder.set_exclude_ring_stiffener(is_excluded=True)
cylinder.set_exclude_ring_frame(is_excluded=False)
cylinder.set_stresses(sasd=-200, smsd=0, tTsd=0, tQsd=5, psd=0, shsd=-60)

print_flat_report(flat)
print_cylinder_report(cylinder)

-----------------------------------------------------------------------------

Example output headings:

FLAT PLATE
Available buckling methods: DNV-RP-C201 - prescriptive, SemiAnalytical S3/U3,
ML-Numeric (PULS based)
DNV-RP-C201 max UF: ...
SemiAnalytical selected UF: ...
SemiAnalytical valid: ...
Plate thickness: actual=..., minimum=...

CYLINDER
Unstiffened shell: ...
Longitudinal stiffened shell: ...
Heavy ring frame: ...
Column stability UF: ...
Stiffener check: ...
```

Startup

At startup user is asked to select either multiple panel mode or single panel mode. When selecting single panel mode the following applies:

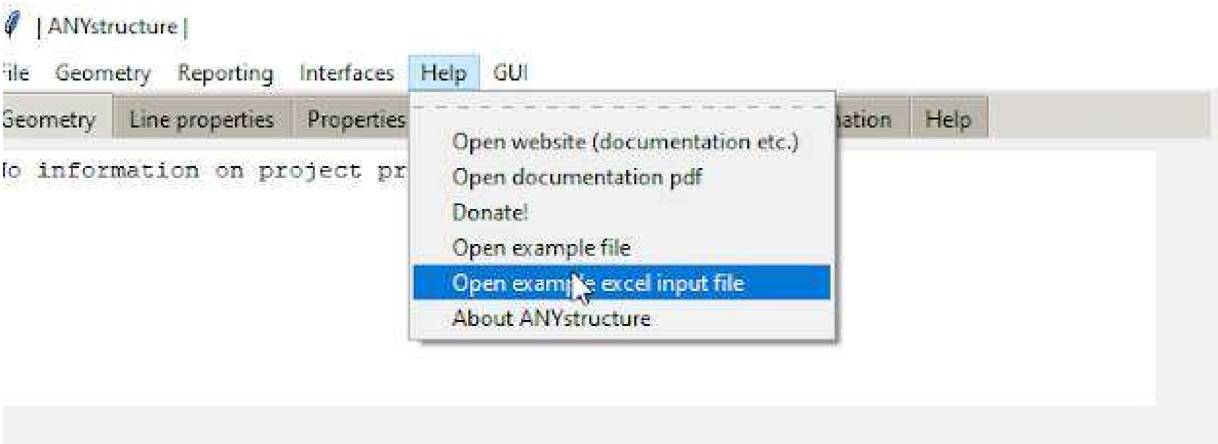
1. 3D representation in the large canvas
2. Pressure input for plat plates simplified to manual input.
3. One line is automatically modelled. Further modelling not required.
4. All other input options are available.



Sheet input

Excel or any other sheet compatible software can be used is input basis for ANYstructure. Both this applies both for flat plates and cylinders. Use “Open excel input” under files. The input is specific.

An excel file containing the correct input option is located in the installation folder and can also be accessed under “Help”



Modeling

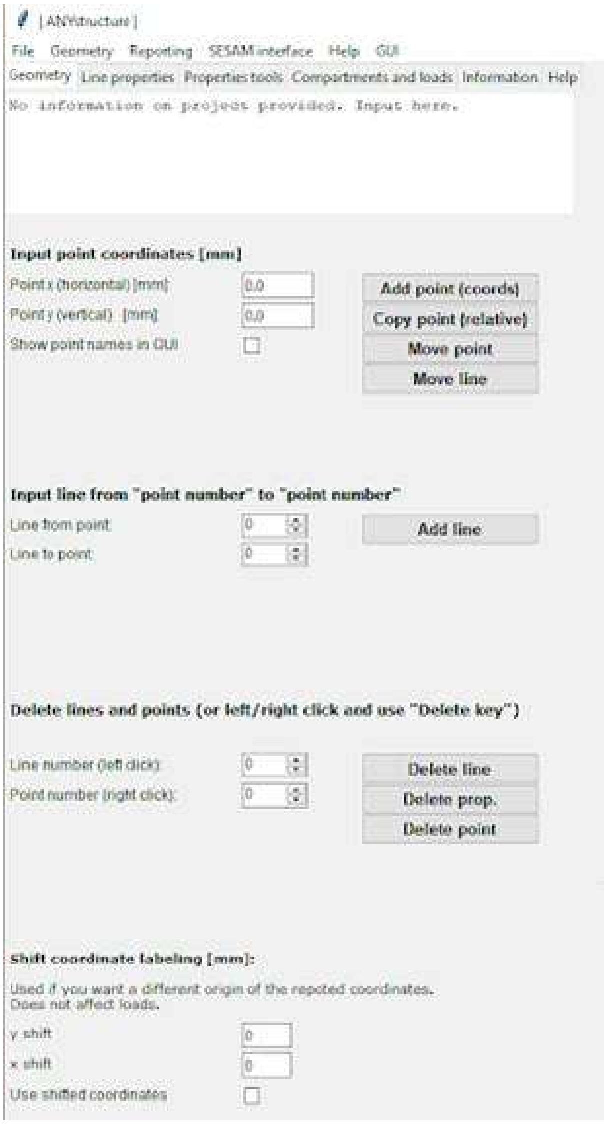
Modeling is done in the Geometry tab.

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)

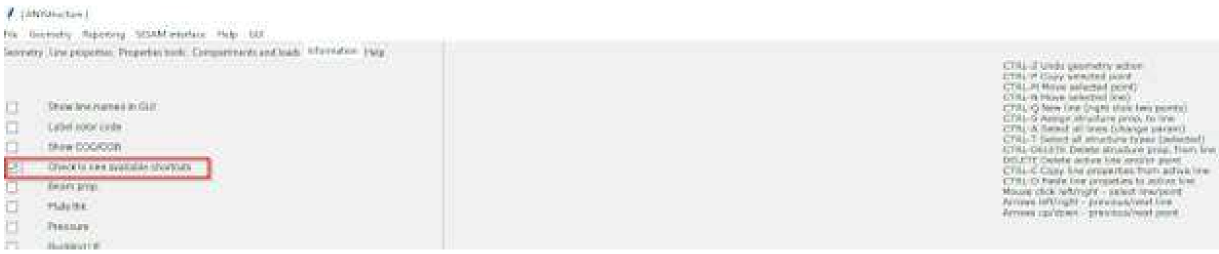


Speed up your modeling significantly by using the shortcuts:

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.

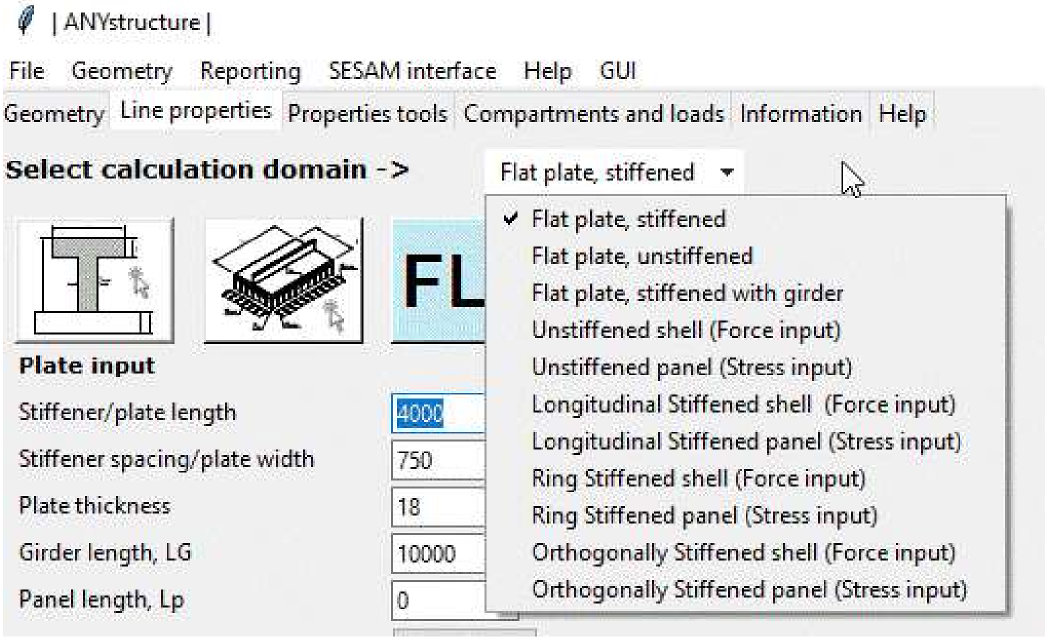


Assigning properties

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.

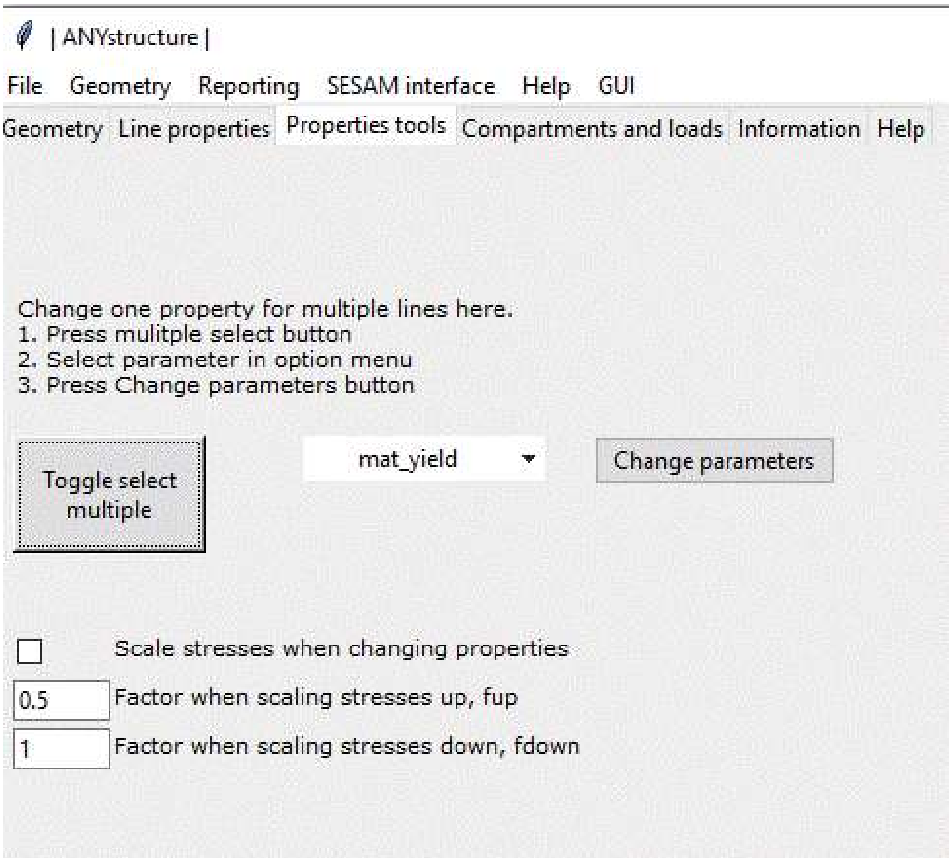
The first value to be set is the structure type. The selection is either a flat plate or some type of cylindrical structure (all others). The dropdown list is shown next. More details on the cylindrical structures.



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 (SigmaX), transversal stress(SigmaY) and shear stress (TauXY).

The parameters fdown = 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

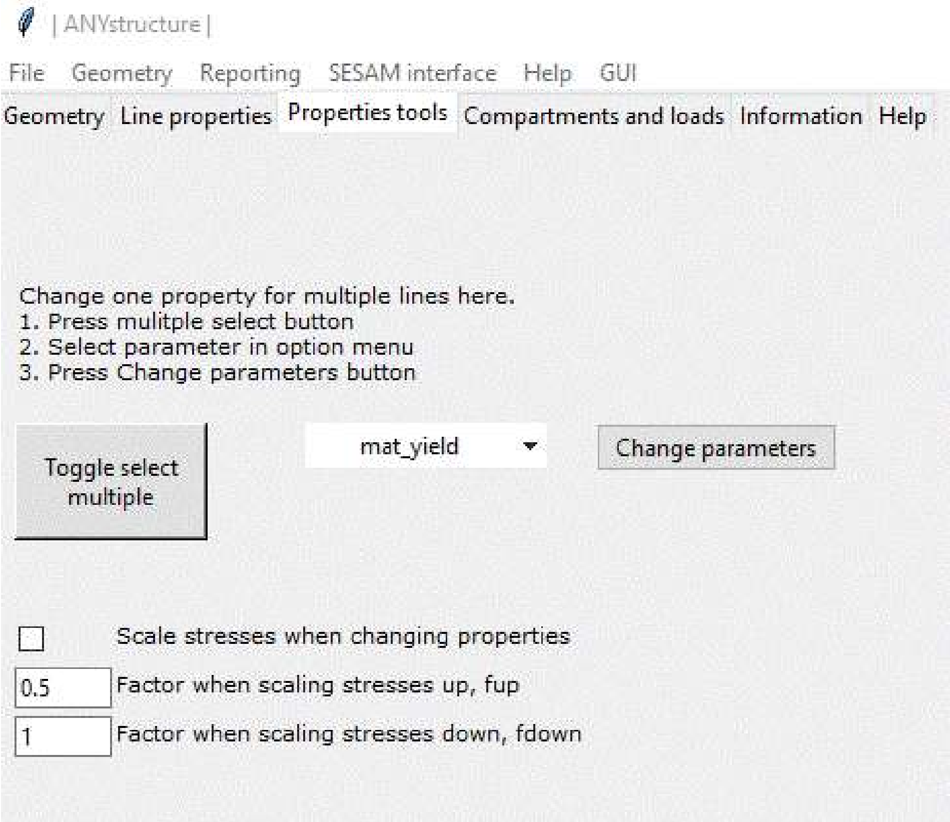
Variables

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

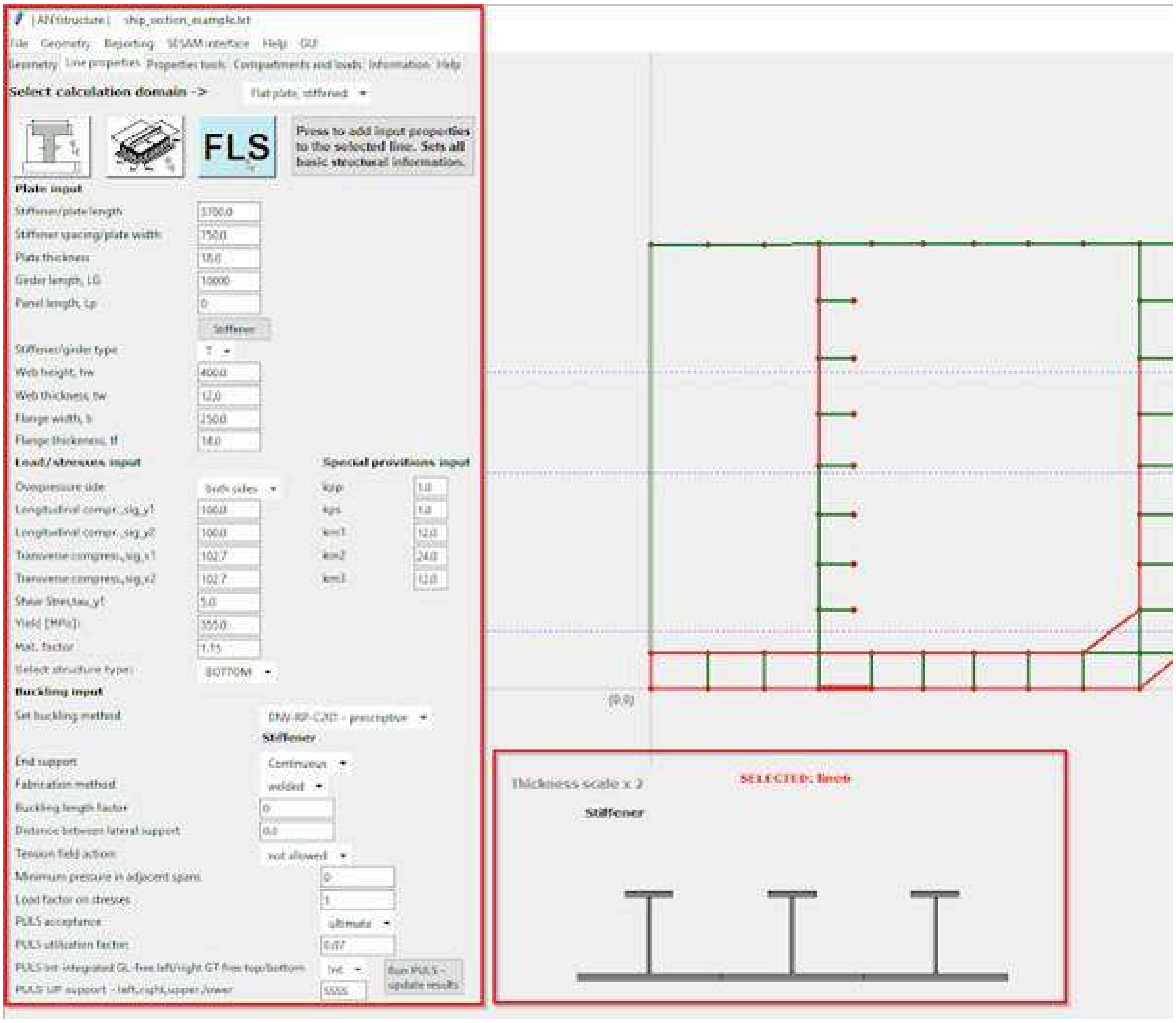
If you want to change a single property for multiple lines. How to do it:

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.

Loads

Manual loads

The simplest way to define loads is to input the stresses and pressures manually. The app provide clearly identified field for this.

In single panel mode, this is the only input for pressure.

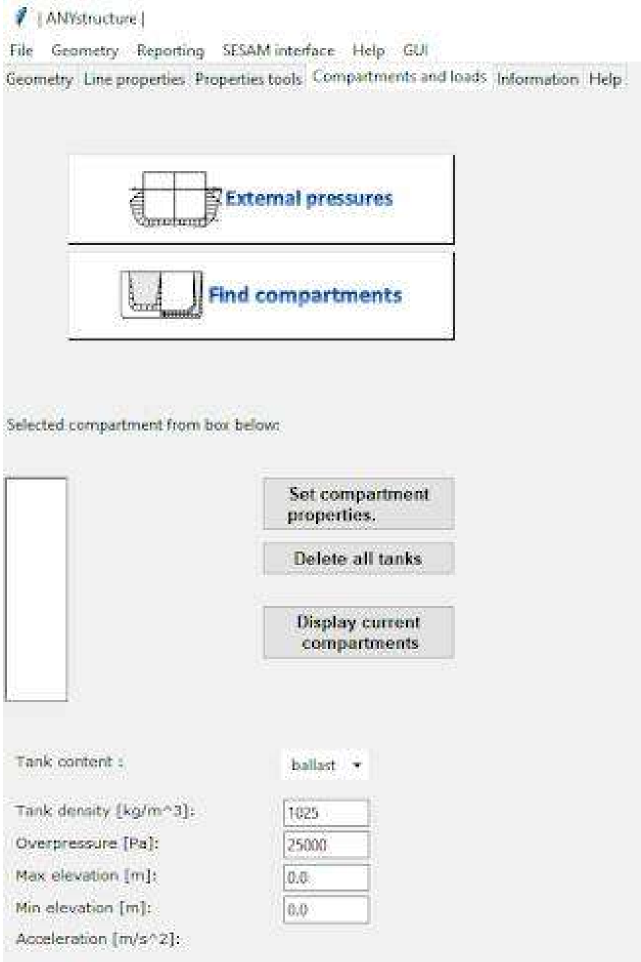
Define tanks

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

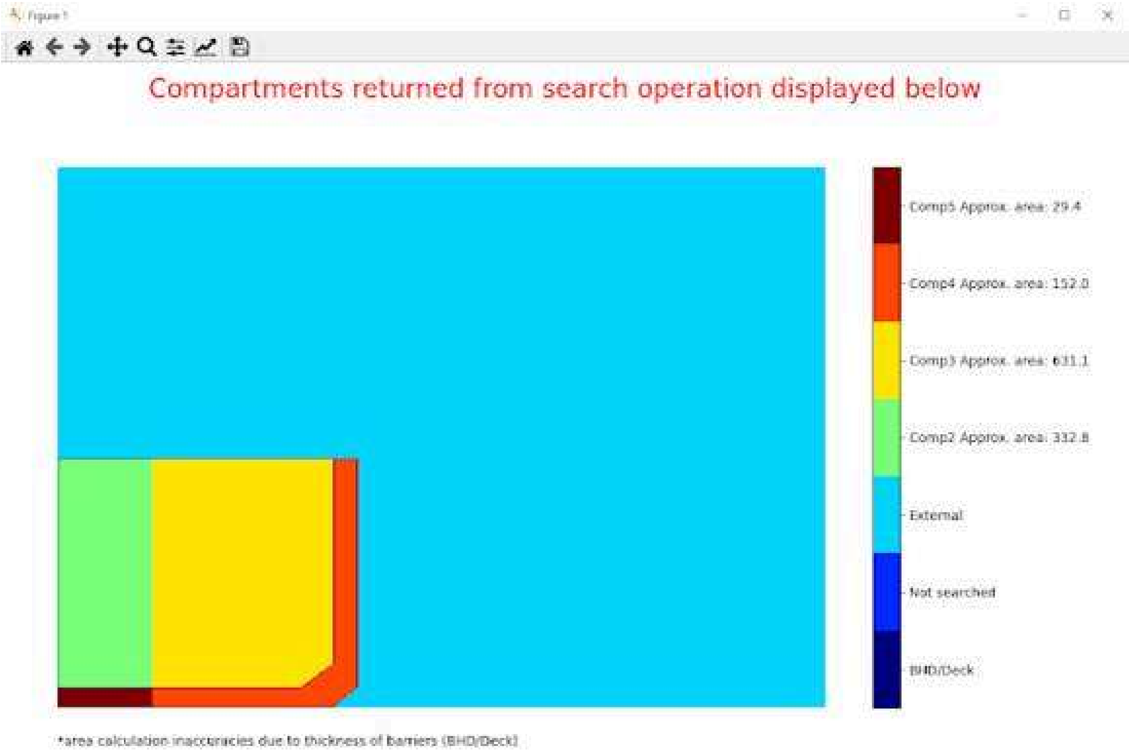
Tanks are searched for when clicking “Find compartments”. Non watertight structures are ignored. For information on structure types click “Show structure types”.

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

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

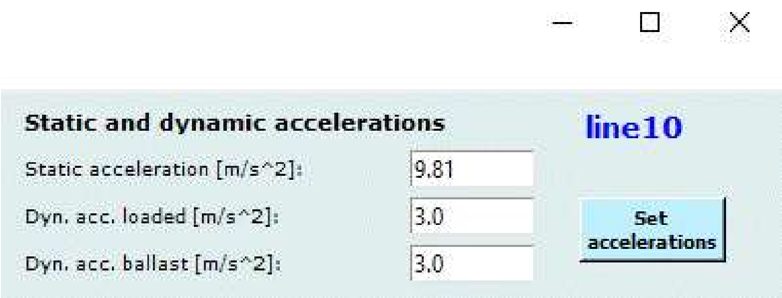


If you press "Display current compartments" after doing a compartment search, the result of the search is illustrated as seen next. Approximate area of the respective compartments is also shown.



Setting accelerations

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



Define external pressures

Click “External pressures” to define pressures acting on the structures.

NOTE:

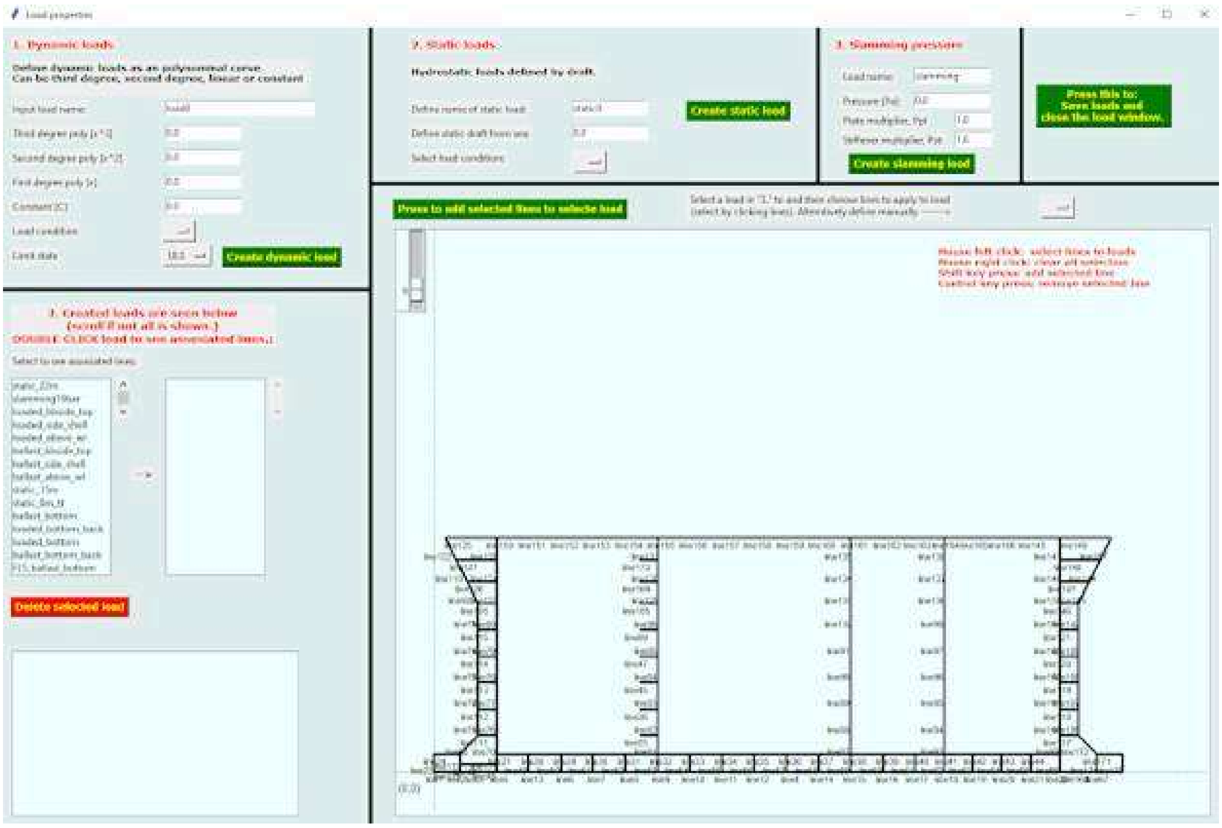
FOR DYNAMIC EQUATION THE FOLLOWING APPLIES

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

Z (vertical) used for BBS, SIDE_SHELL, SSS

After new window is opened:

1. Make dynamic loads
    1. Dynamic loads are made by defining up to 3rd degree equations. X or Y direction depends on the defined structure type.
    2. 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:
    1. 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.



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:

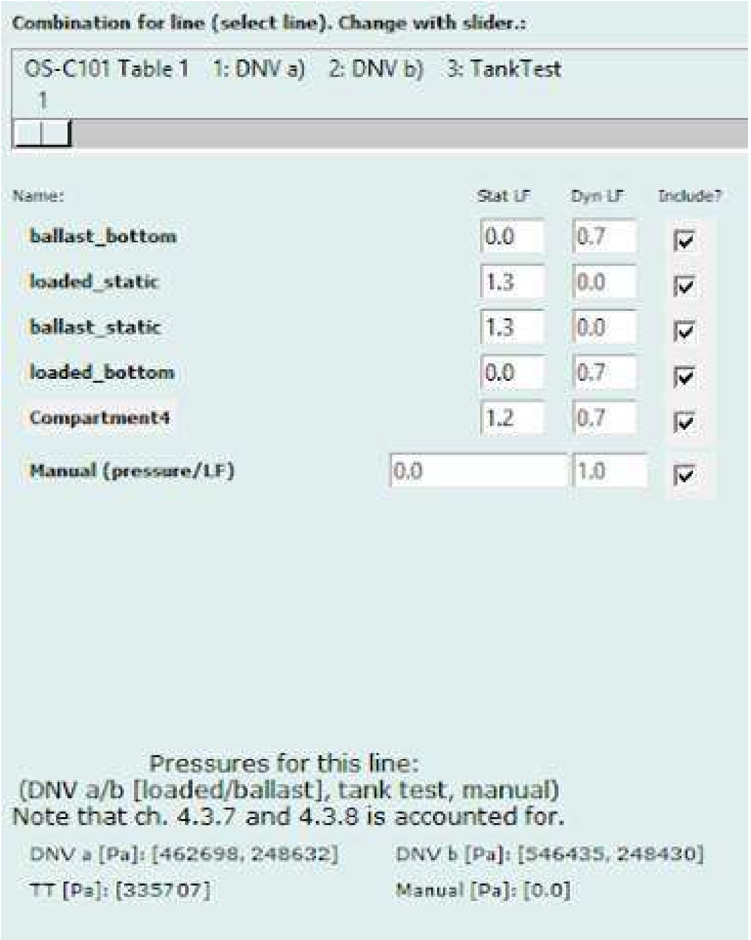
Pplate = 10 bar * factor_plate

Pstf = 10 bar * factor_stiffener

Load combinations

Load combinations are created automatically after external pressures are defined. Some comments on the loads.

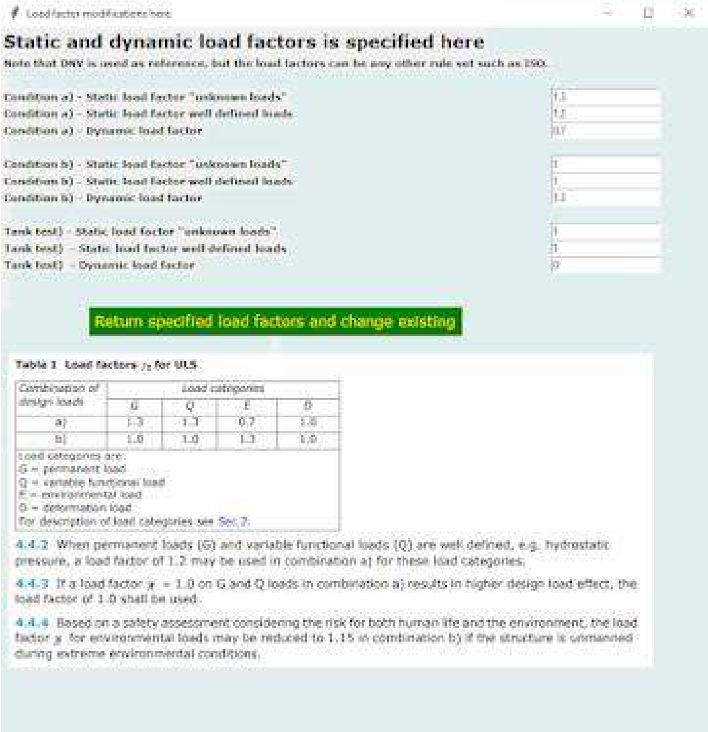
1. According to DNVGL-OS-C101
2. Highest pressure is chosen w.r.t. tank filling.
3. You can deselect a load by manually inputting load factor to 0 or deselect include.



Changing load factors

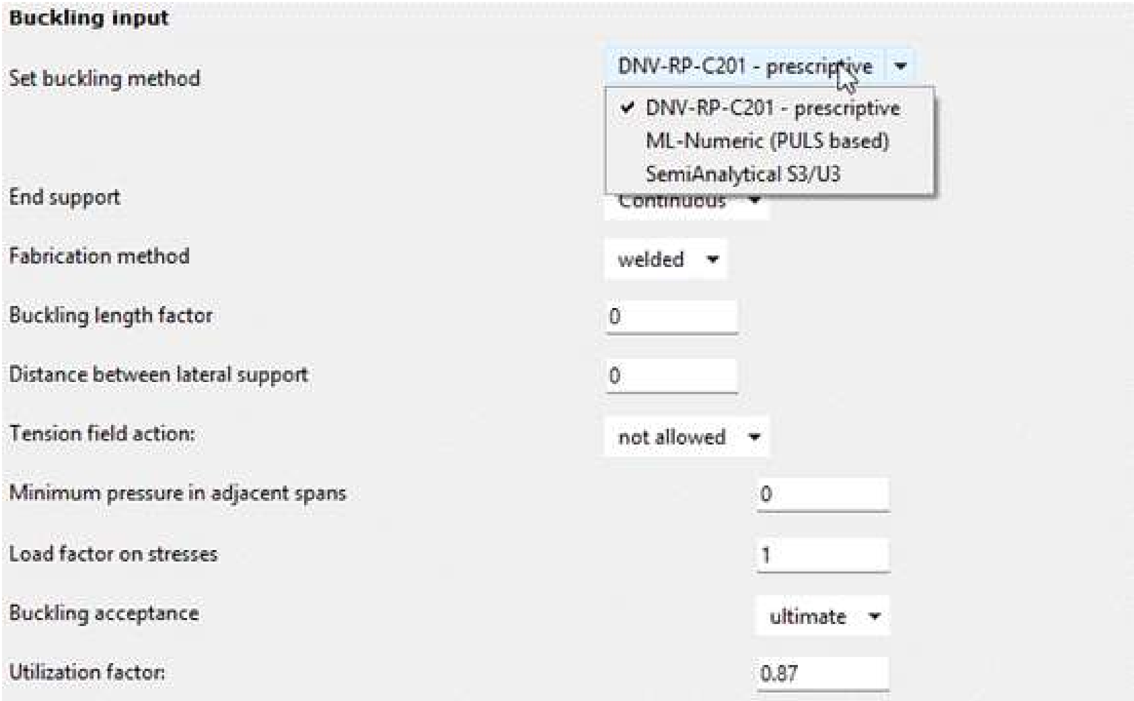
You can change default load factors and existing load factors using the button seen in the next illustration.

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



Buckling

ANYstructure has 3 options for calculating buckling.



Semi-Analytical Buckling

Semi-analytical buckling is used to evaluate plate and stiffened-panel buckling based on established analytical and semi-empirical formulations. The method provides a fast alternative to detailed finite element buckling analysis and is suitable for design iterations and optimization.

The semi-analytical buckling options use the same structural input as PULS, including plate geometry, stiffener properties, material data, boundary conditions, and applied loads. Depending on the selected option, the buckling response may be calculated directly using semi-analytical formulations or estimated using a trained machine-learning model based on numerical PULS results.

ML Buckling

The ML buckling option is a neural-network-based approximation of numerical PULS results. It uses the same input parameters as PULS, but the PULS utilization factor is not applied.

The classification neural network is currently deactivated. ML buckling therefore uses only the numerical neural-network prediction.

The results should be used with care, as neural-network predictions may occasionally be incorrect. The current prediction accuracy is approximately 97%.

ML buckling can be used with all optimization options.

Cylinder buckling

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

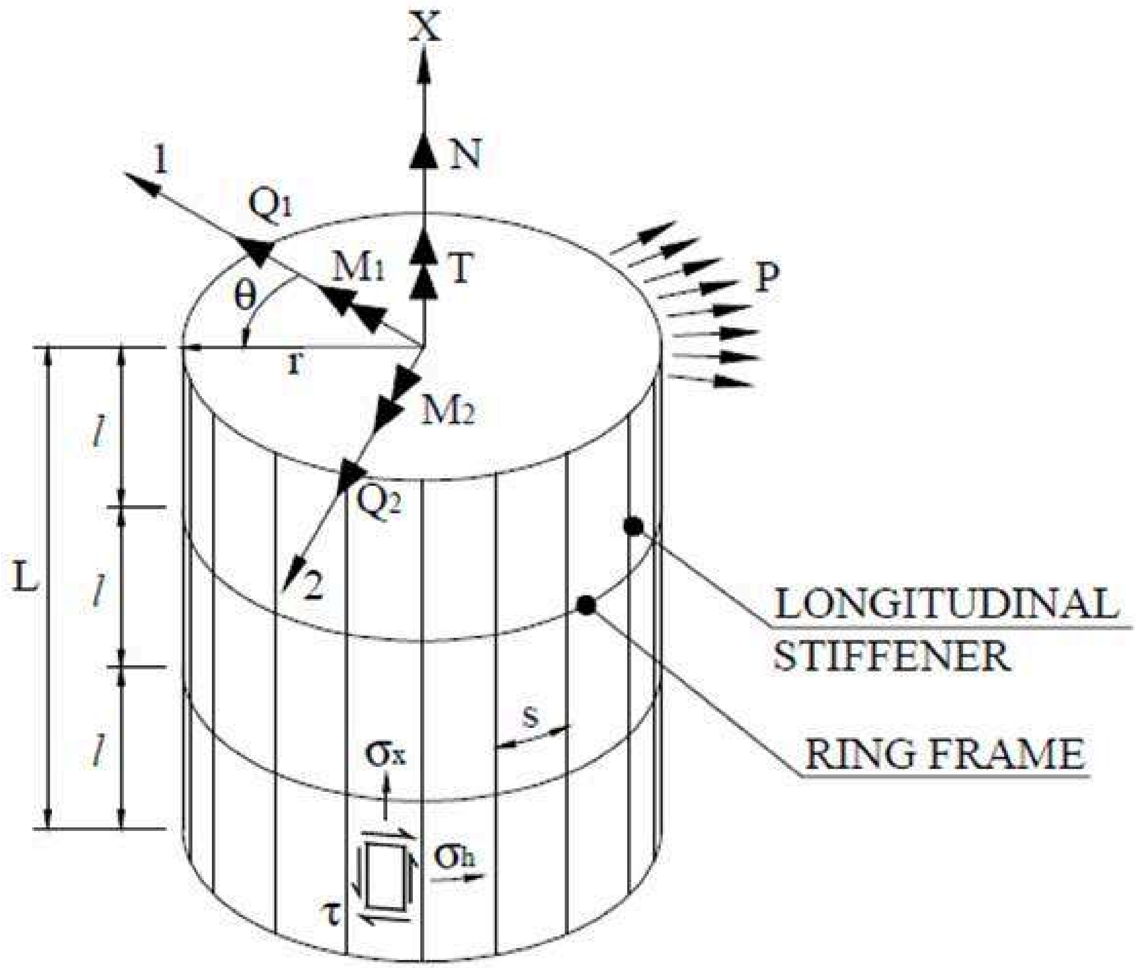
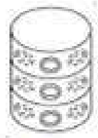
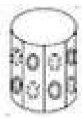
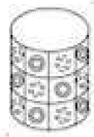
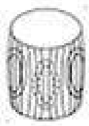
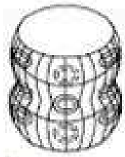
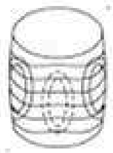
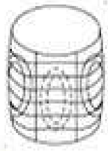
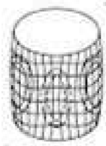
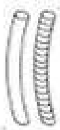
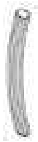


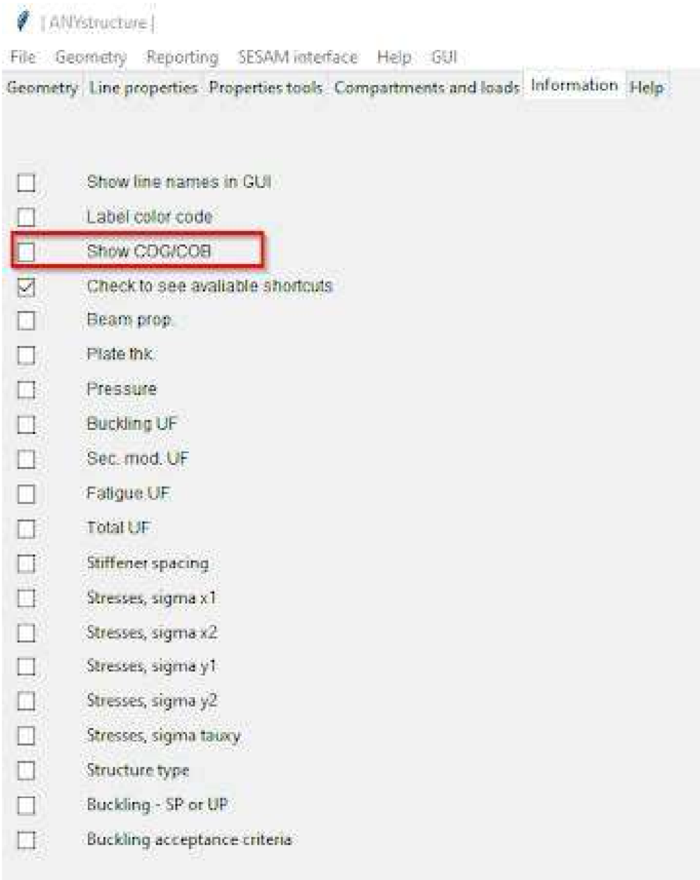
Table 1-1 Buckling modes for different types of cylinders

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

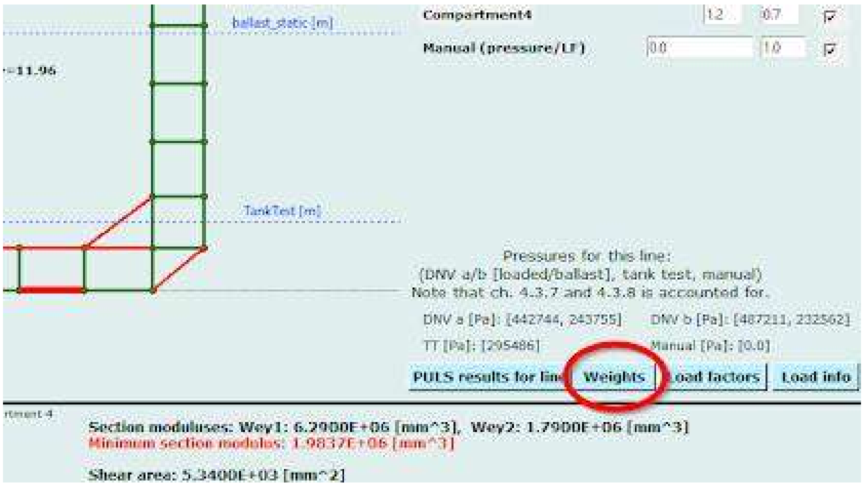
Reviewing data

COB and COG

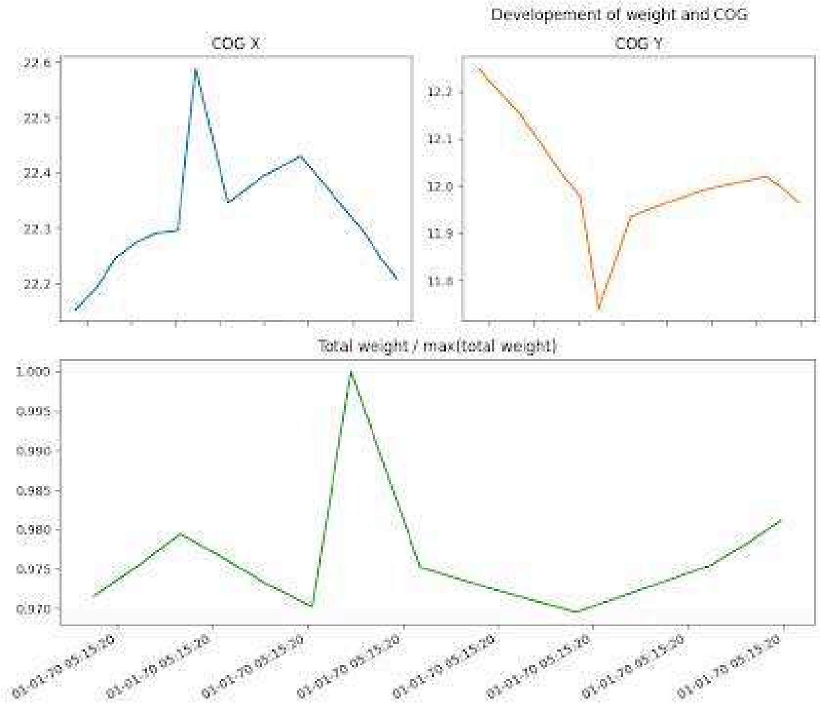
COG is calculated when structure properties are defined. COB for various drafts is calculated when tank search is completed.



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



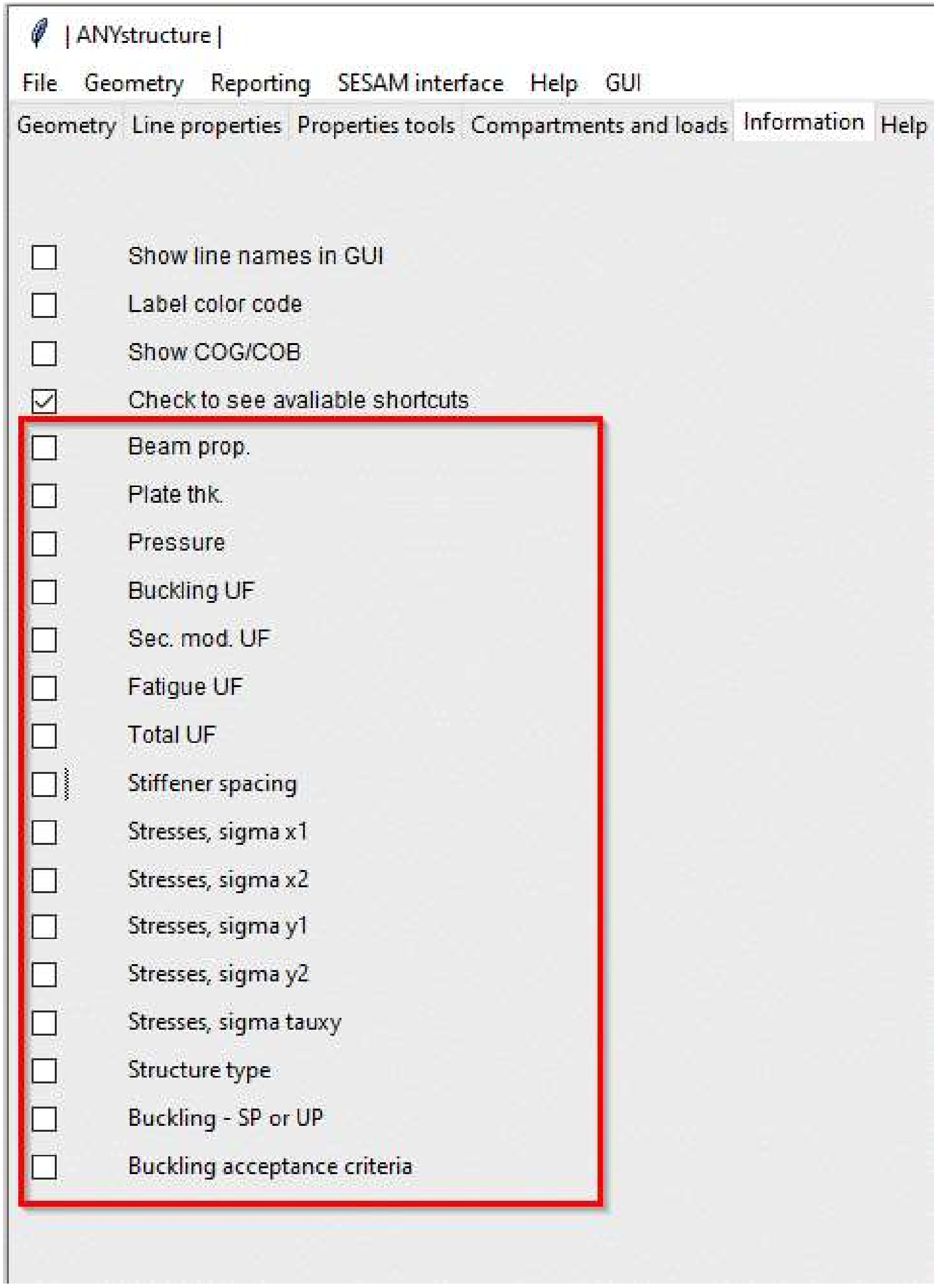
Output example is seen next.



Plates in model	Beams in model
18.0 mm	T_400_0x12_0__200_0x20_0
25.0 mm	T_400_0x12_0__250_0x14_0
12.0 mm	T_400_0x12_0__250_0x12_0
20.0 mm	T_400_0x12_0__200_0x18_0
14.0 mm	T_400_0x12_0__150_0x20_0
30.0 mm	T_500_0x12_0__150_0x20_0
15.0 mm	T_340_0x12_0__200_0x20_0
	T_340_0x12_0__150_0x16_0
	T_250_0x12_0__150_0x14_0
	T_450_0x12_0__150_0x20_0
	T_375_0x12_0__150_0x18_0
	T_500_0x12_0__150_0x25_0
	T_325_0x12_0__150_0x16_0
	FB_250_0x18_0
	FB_400_0x18_0
	T_350_0x12_0__150_0x20_0
	T_320_0x12_0__150_0x20_0
	T_300_0x12_0__150_0x20_0

Color coding

All color coding options are indicated next.

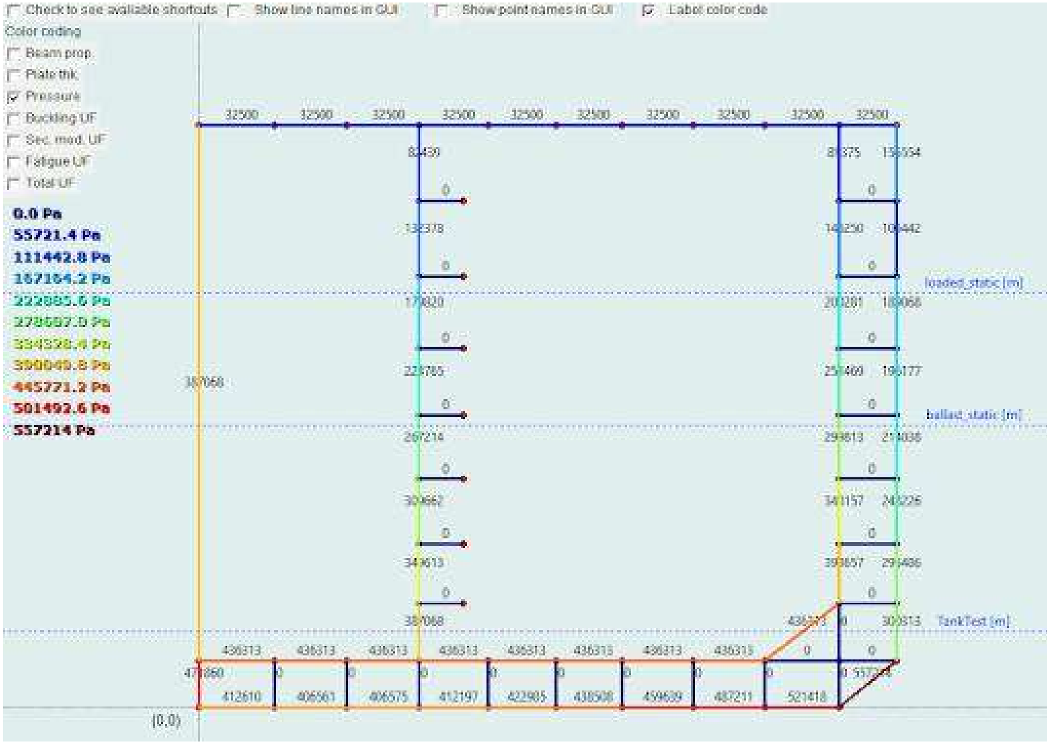


Shifting coordinates for visual purposes

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.



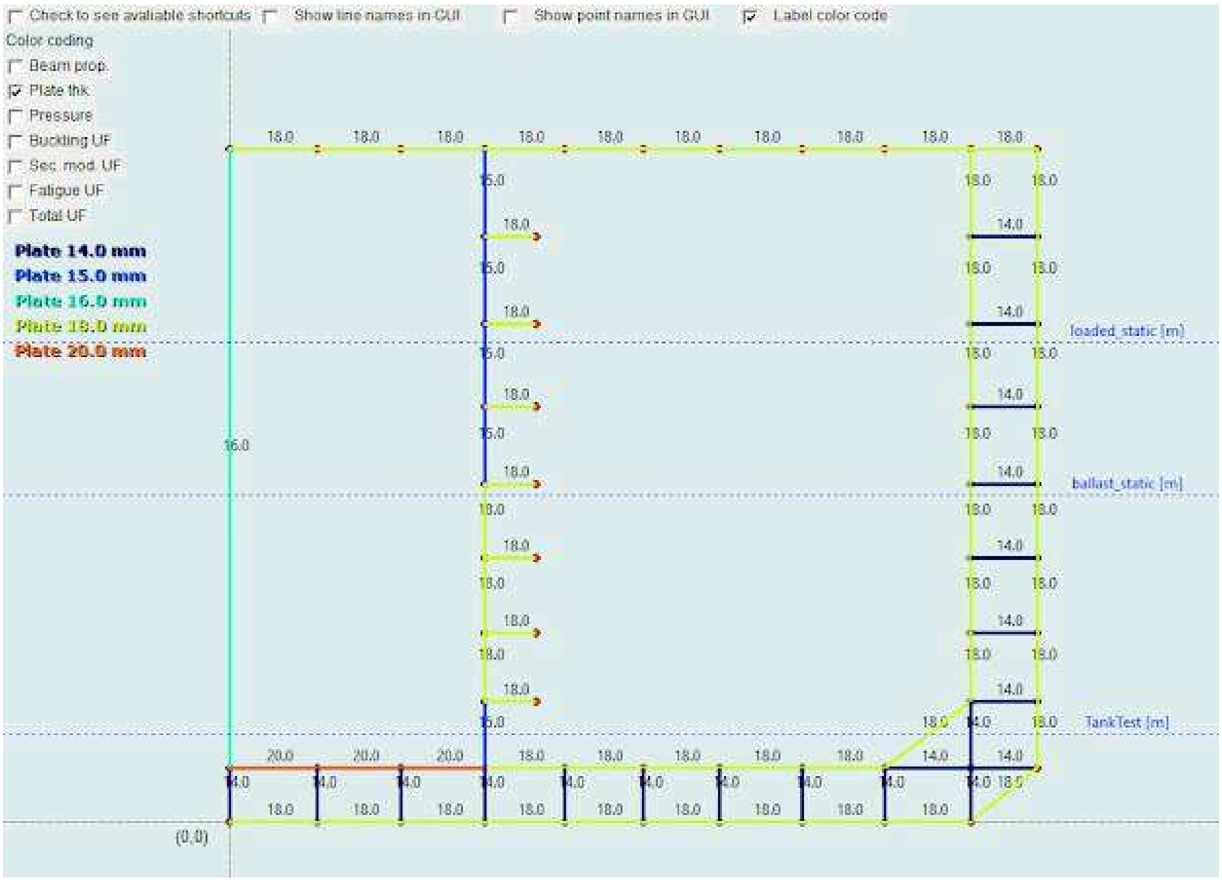
Load calculations and results can be reviewed by clicking the “Load info” button. An example is seen in the next illustration.

| ANYstructure | ship_section_example.txt

```
Load results for line8

Loads for condition: loaded - dnva
 static with acceleration: 9.81 is:
 1*1.3*221215.5 = 287580.2
 dynamic with acceleration: 3.0 is:
 1*0.7*181077.2 = 126754.1

RESULT: 287580.2 + 126754 = 414334.2
------------------------------------------------------------------
Loads for condition: ballast - dnva
 dynamic with acceleration: 3.0 is:
 1*0.7*57425.2 = 40197.6
 static with acceleration: 9.81 is:
 1*1.3*150828.8 = 196077.4

comp4 - static: 1*1.2*310707.22500000003 + 25000.0*1.3 = 405348.67000000004
comp4 - dynamic: 1*0.7*95017.50000000001 + 25000.0*0 = 66512.25

RESULT: 40197.6 + 196077 = 236275.0
------------------------------------------------------------------
Loads for condition: loaded - dnvb
 static with acceleration: 9.81 is:
 1*1.0*221215.5 = 221215.5
 dynamic with acceleration: 3.0 is:
 1*1.2*181077.2 = 217292.7

RESULT: 221215.5 + 217293 = 438508.2
------------------------------------------------------------------
Loads for condition: ballast - dnvb
 dynamic with acceleration: 3.0 is:
 1*1.2*57425.2 = 68910.2
 static with acceleration: 9.81 is:
 1*1.0*150828.8 = 150828.8

comp4 - static: 1*1.0*310707.22500000003 + 25000.0*1.3 = 343207.22500000003
comp4 - dynamic: 1*1.3*95017.50000000001 + 25000.0*0 = 123522.75000000003

RESULT: 68910.2 + 150829 = 219739.0
------------------------------------------------------------------
Tank test for: t
1 * 1.0 * 40221.0 + 0 = 40221
Tank test for: comp4
1 * 1.0 * 310707.2 + 25000.0 * 1 = 335707
Manual pressure:
0.0 * 1.0 * 1 = 0.0
```

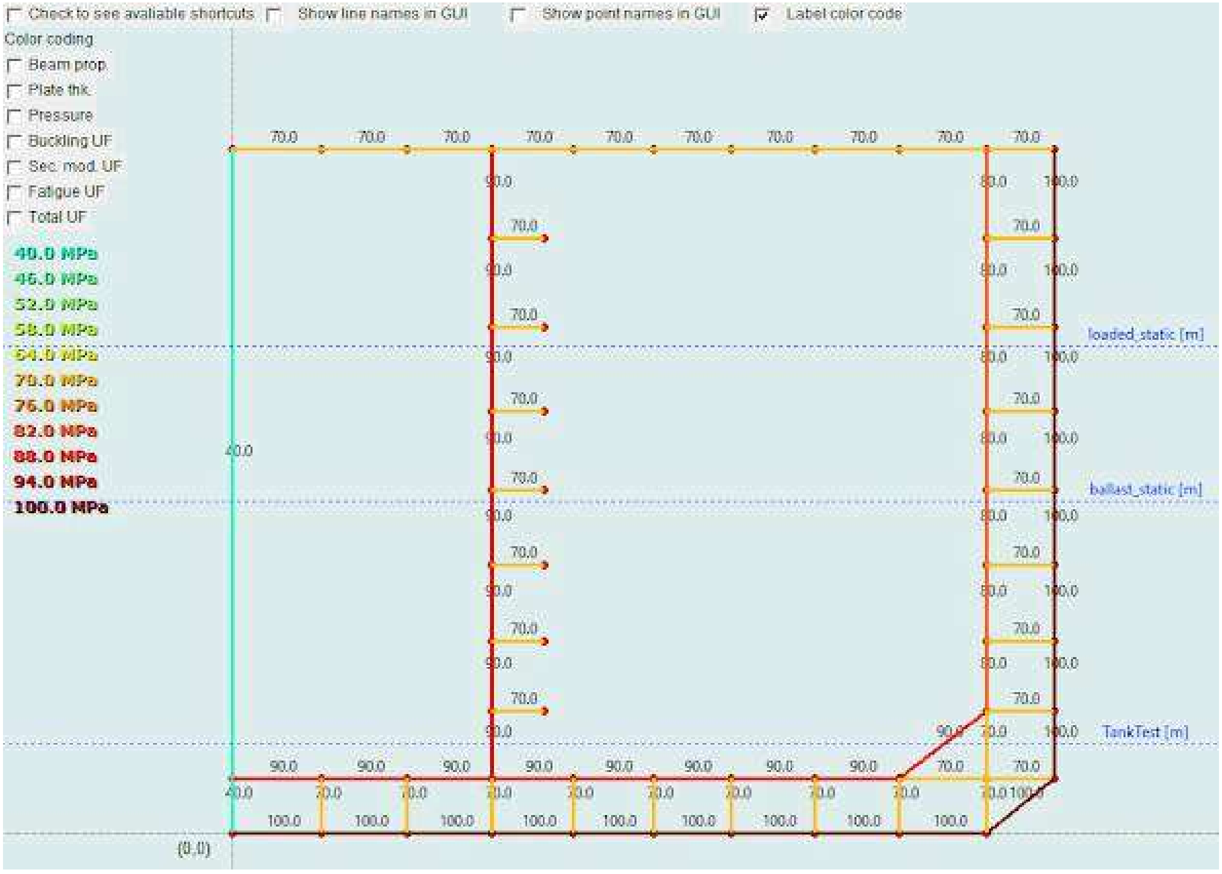
Thickness and beam properties

Line plate thicknesses and beam properties can be reviewed using color coding. Plate thicknesses are exemplified next.



Global stresses (buckling) and structure types

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


Check to see avaliable shortcuts
Show line names in GUI
Show point names in GUI
Label color code
Color coding
Beam prop.
Plate thk.
Pressure
Buckling UF
Sec. mod. UF
Fatigue UF
Total UF
40.0 MPa
46.0 MPa
52.0 MPa
58.0 MPa
64.0 MPa
70.0 MPa
76.0 MPa
82.0 MPa
88.0 MPa
94.0 MPa
100.0 MPa
loaded_static [m]
ballast_static [m]
TankTest [m]
(0,0)

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.

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

	Minimum value	Actual value	Accepted?
Section modulus check	1.8617E+06 [mm^3]	1.9600E+06 [mm^3]	Ok
Shear area check	3.6307E+03 [mm^2]	5.2320E+03 [mm^2]	Ok
Plate thickness check	14.4 [mm]	18.0 [mm]	Ok

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

	Plate	Stiffener	Girder
Overpressure plate side	0.574	1.244	0
Overpressure stiffener side		1.112	0
Resistance between stiffeners		0.832	0
Shear capacity		0.684	
Maximum web height [mm]		410.0	0
Maximum flange width [mm]		205.0	0

Fatigue results (DNVGL-RP-C203):
Total damage (DFF not included): 0.137 | With DFF = 2.0 --> Damage: 0.275

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

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

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

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

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

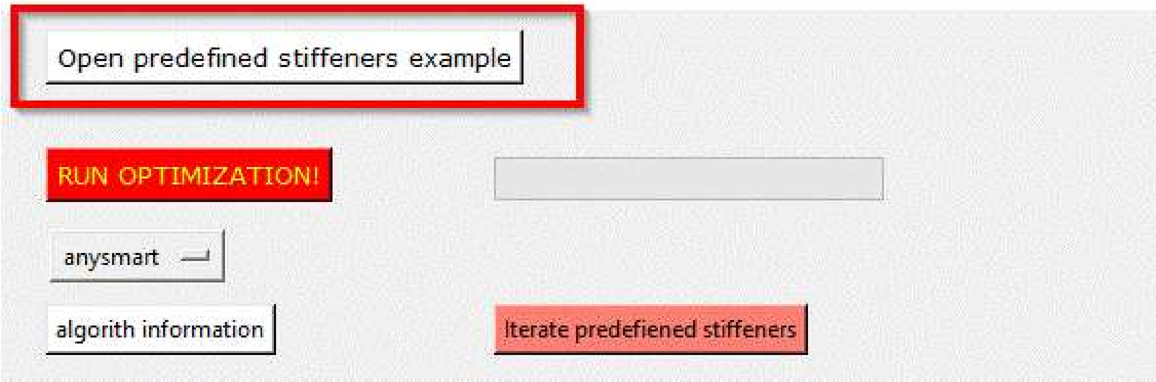
A combined utilization can be reviewed using color coding.

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.

Single optimization

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

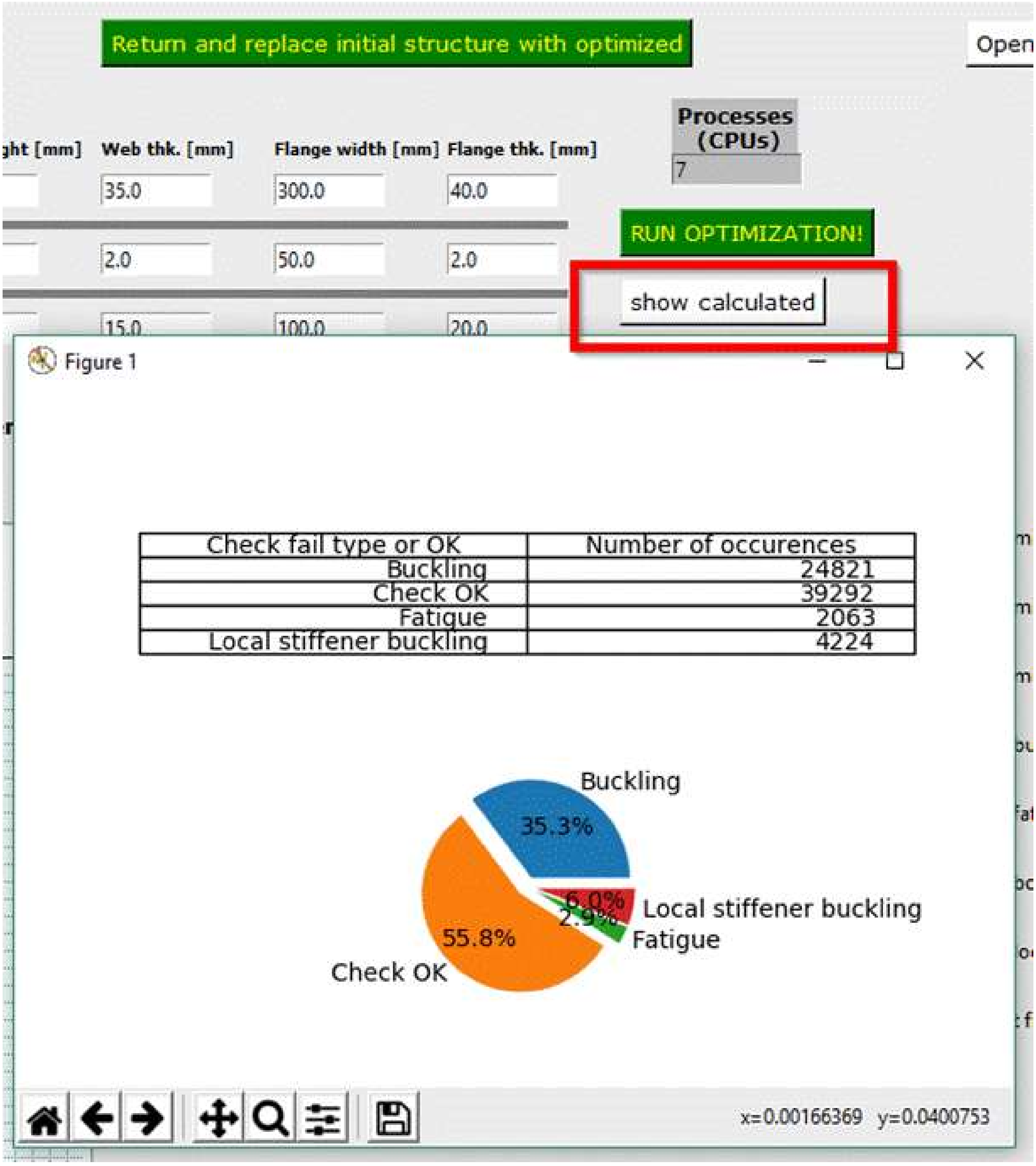
1. Set the upper and lower bounds of the optimization.
2. Set the delta to be used for the search. This is the step size of the optimization when using brute force method (for example anysmart).
3. Run the optimization.
4. If you are happy, return the properties by clicking the top button

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.

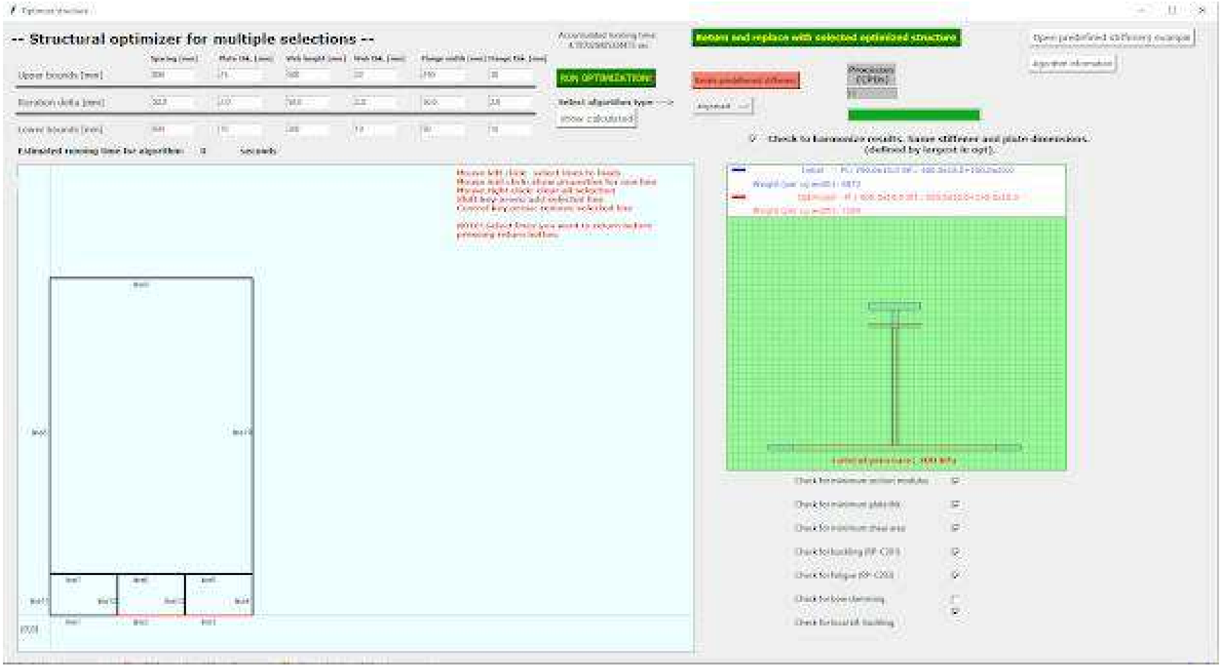
The weight filter ensures that only sections with a lower weight than the current minimum weight. This significantly speeds up the calculations, but if you want to see the full distribution of the various checks this must be unchecked.

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 “occurence” is a one checked plate/stiffener combination.



You will also be asked to save to a csv file. If you do not cancel, a csv file will ALL results will pre saved to your chosen location. If you open the file in excel you should see something like show next

Multiple optimization



Multiple optimization is done by clicking the “MultiOpt” button.

1. Same input on upper bounds, lower bounds and delta.
2. Click all the lines you want to include in the optimization.
3. Run the optimization.
4. Check the properties by middle clicking the line you ran.
5. If you are happy, return the properties by clicking the top button. Remember to select the lines you want to return. Lines that have been optimized are marked orange.

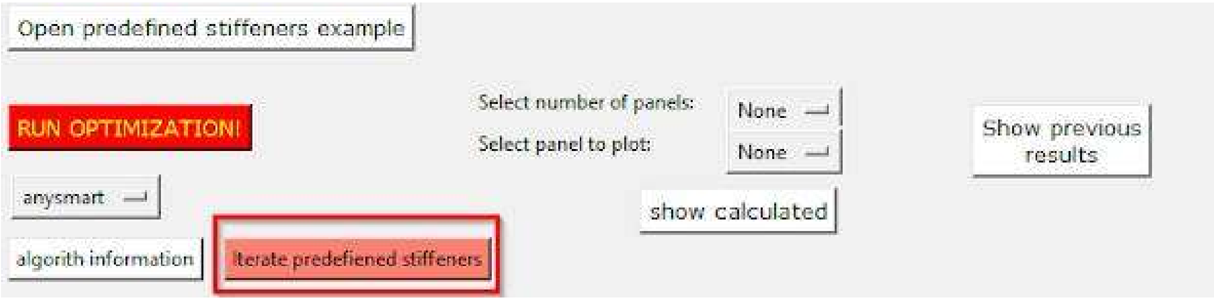
The optimization can be harmonized. That means that the largest dimension found in the multiple optimization is used for all selected. This is done after all plates/stiffeners are checked. Harmonization can only be done in the multiopt option. Note that the weight filter is not used when harmonizing, i.e. running will take some more time.

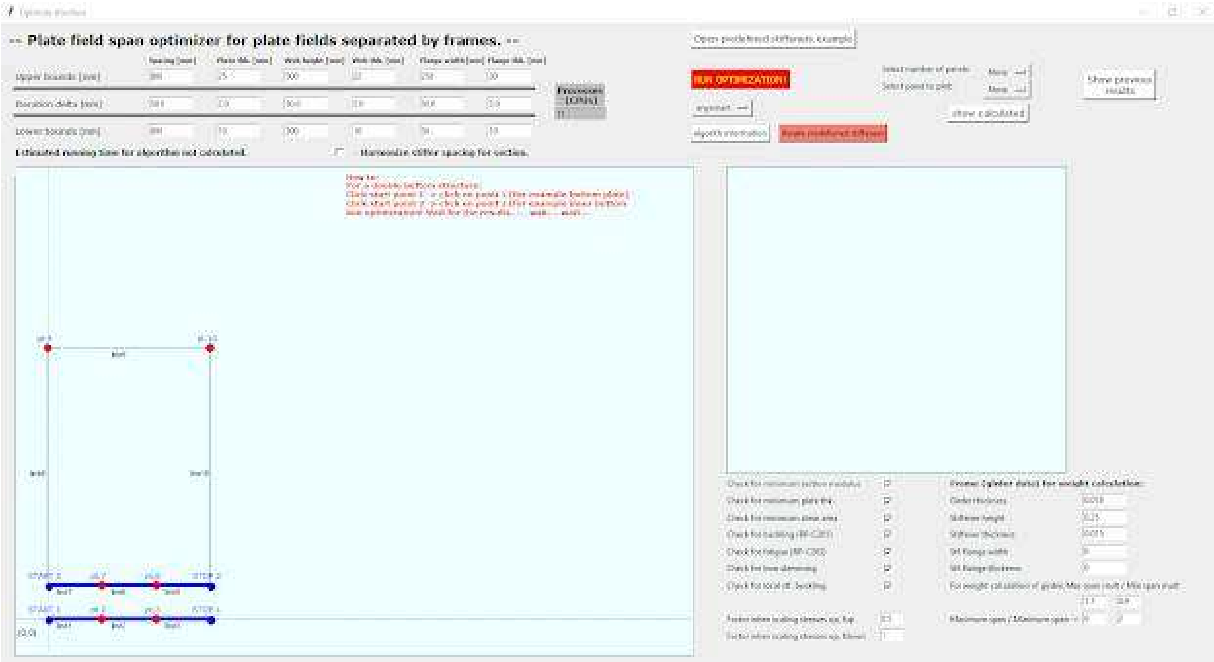
Other options that can be set are explained in the single optimization chapter.

When showing calculated you must have selected a line (middle click).

Span optimization

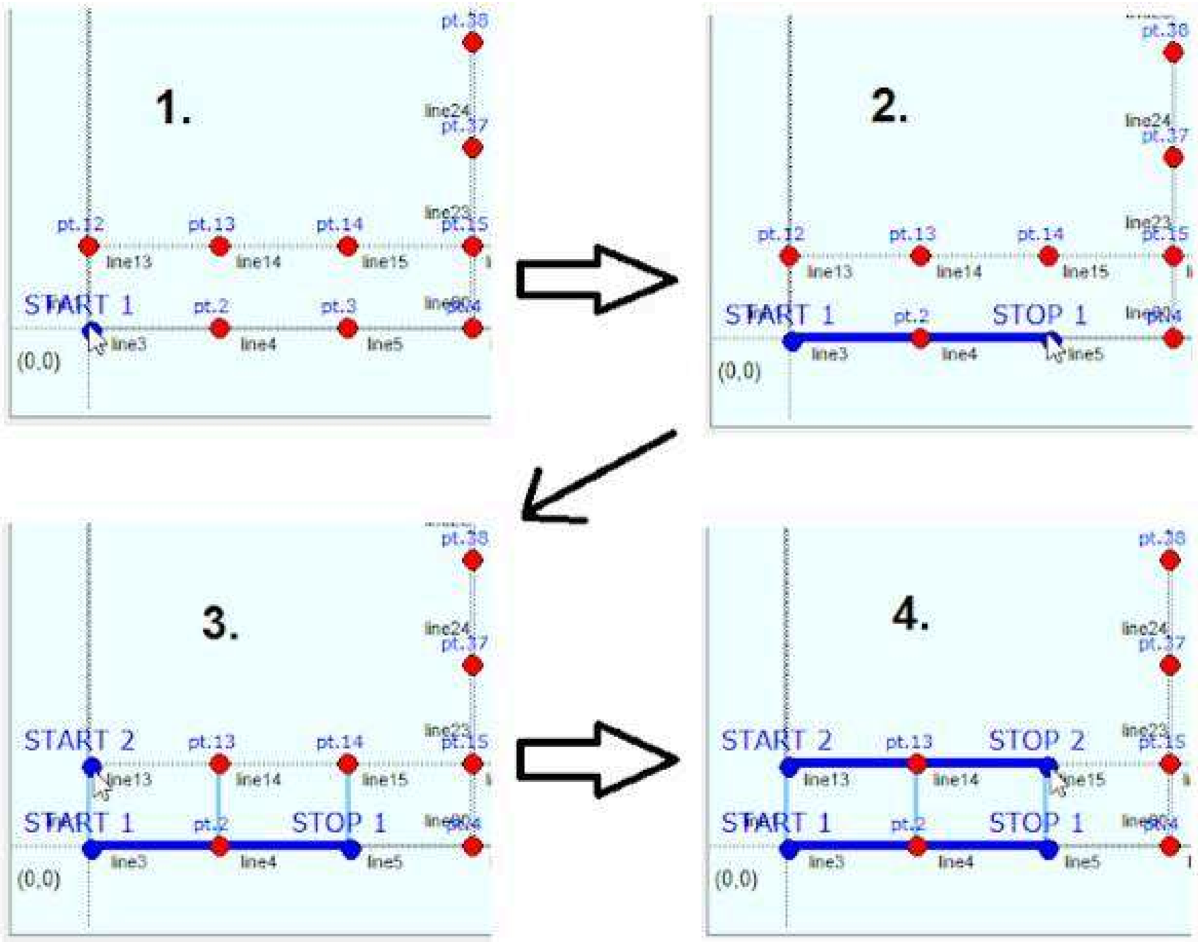
NOTE: The span optimization is computationally heavy. It is recommended to use a set of predefined stiffeners.





The optimization is started as follows.

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

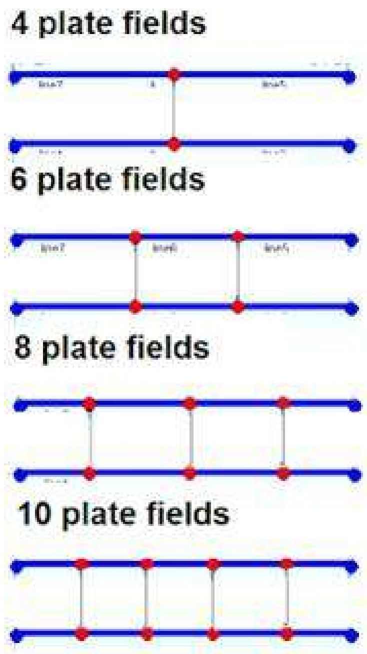


2. Check the input and checkboxes in the lower right corner

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.

3. Start the calculation

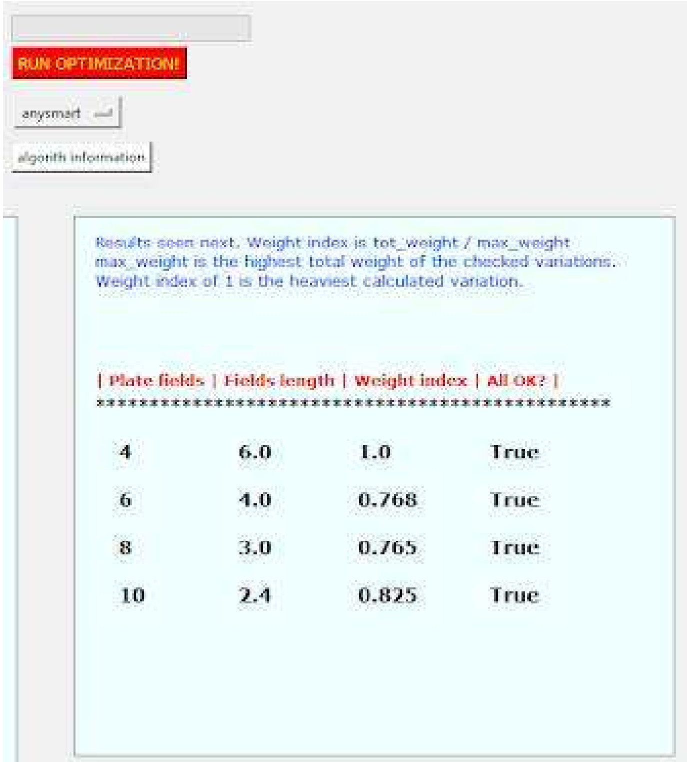
The program will calculate variations of even spans in your structure as illustrated next. This is an example and the number of plate fields may vary.



With reference to the previous example, max span mult is the multiplicator for the 4 plate fields set up and min span mult is the weight multiplication for the 10 plate field set up. This is adopted because one can assume the required dimensions for the girder will reduce when more girders are added.

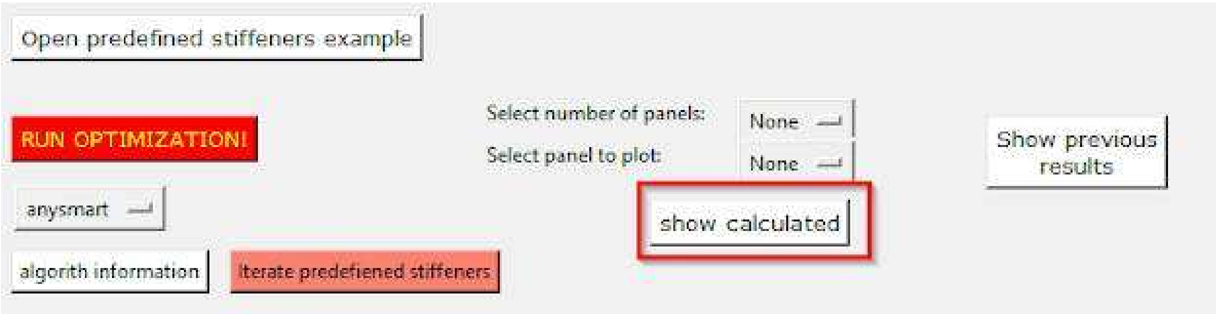
Minimum span and maximum span is the minimum and maximum span of the plate fields in meters.

Results are presented as seen next.



In this case 8 plate fields with length of 3 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.



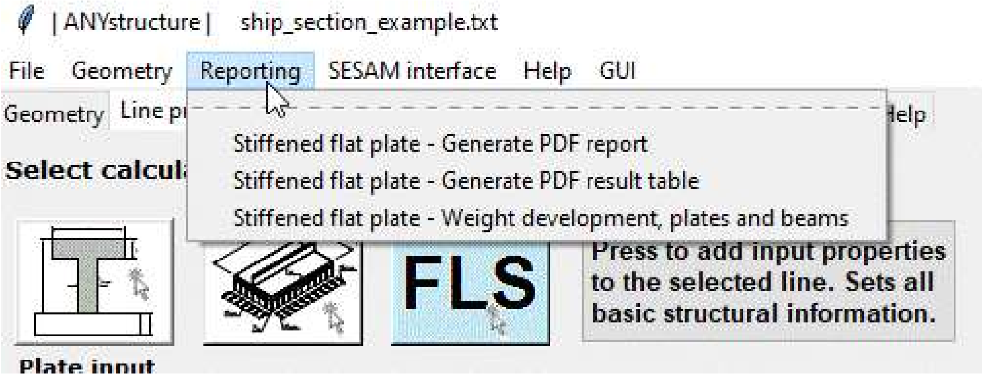
Now close the window. Results are not currently returned to the main window.

Detailed results, printed after running, looks like this :

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

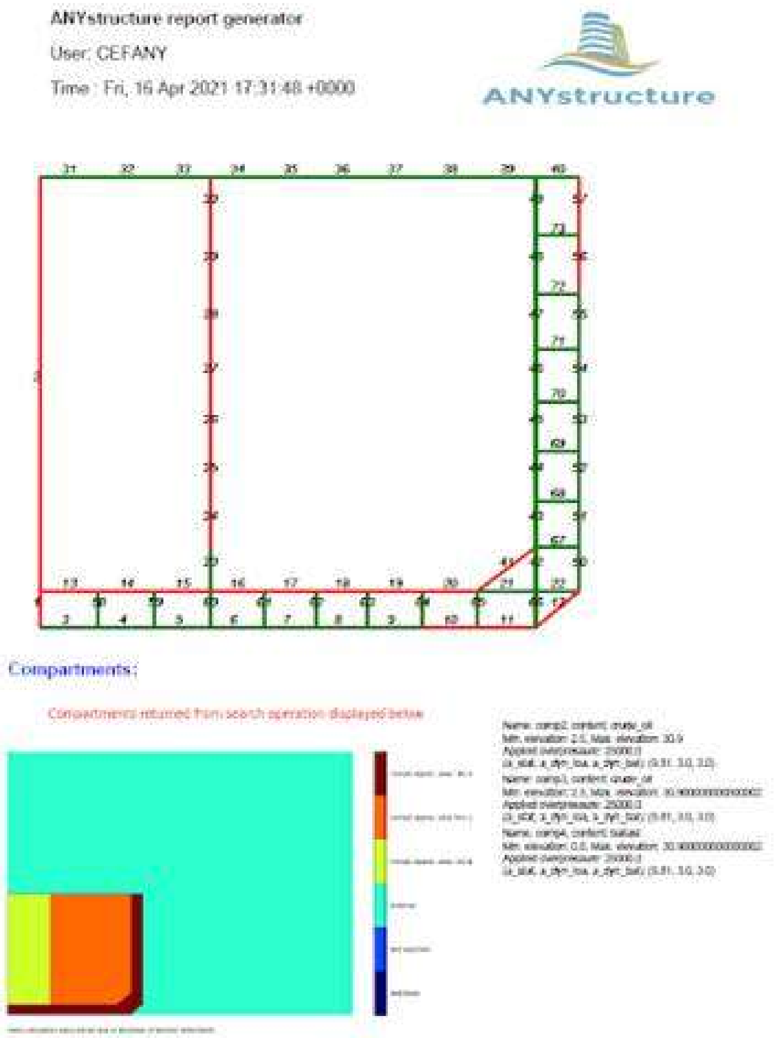
Output and reporting

Report are available as seen below.



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.



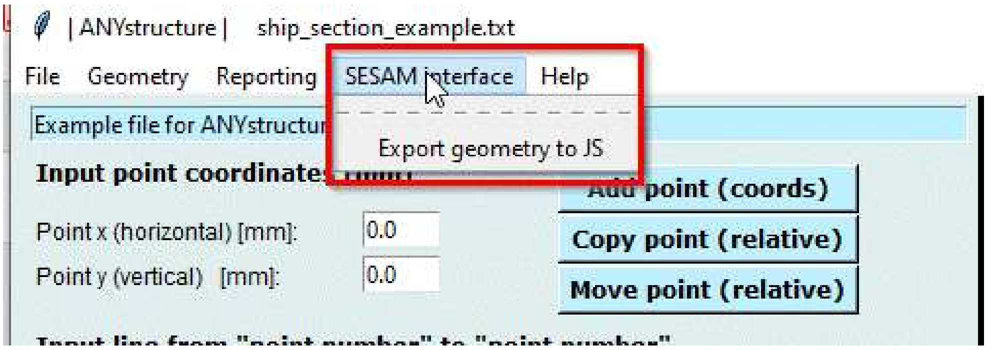
Table

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

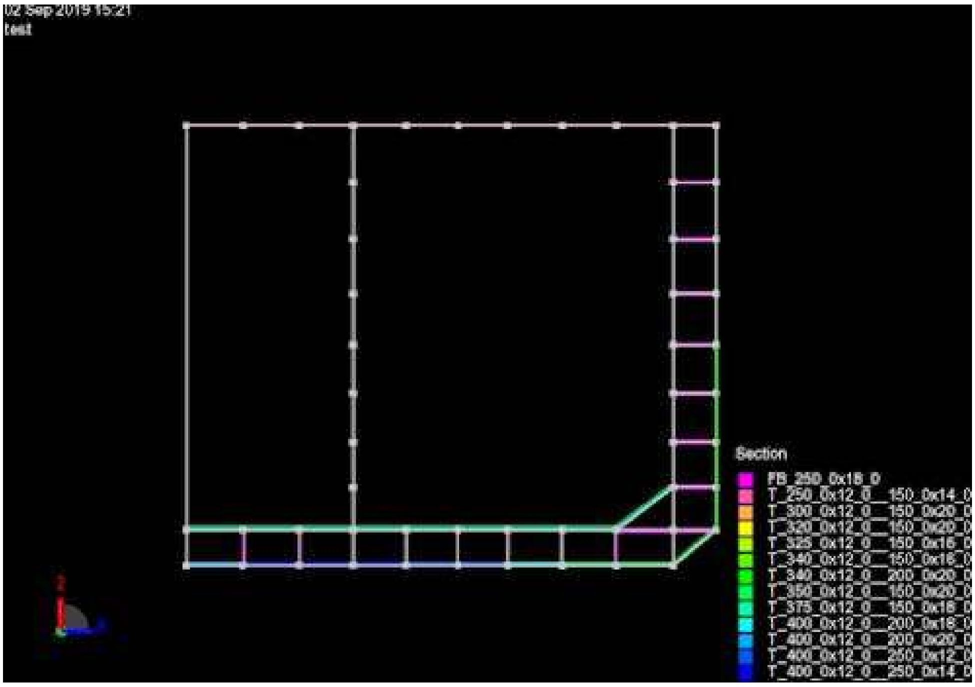
Line	pl thk	s	web h	web thk	fl. w	fl. thk	sig x	sig y1	sig y2	tau xy	max press.	sec. mod	min sec.	min pl	shr area	min shr A	fat uf	buc uf
line1	14.0	700.0	250.0	18.0	150.0	20.0	20.0	40.0	40.0	5.0	472000.0	1036266	630101	13.67	5111	2322		0.32
line10	18.0	750.0	400.0	12.0	150.0	20.0	50.0	100.0	100.0	5.0	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	62000.0	1195190	350252	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.



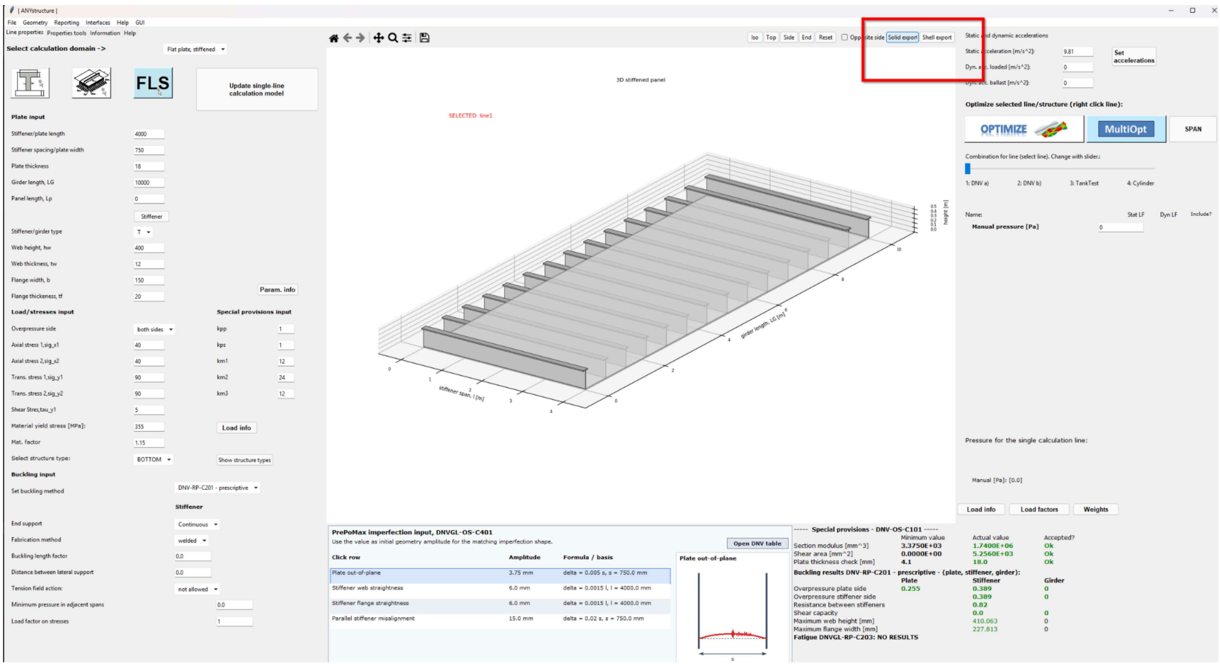
The result is illustrated next:



Exporting 3D

ANYstructure can export plate fields and cylinders to either **solid or shell** 3D CAD formats. The export button are shown in the next figure.

These models can be used as input to FEM analysis. If you wish to check non-linear buckling, the recommended imperfection is shown in the lower-mid part.



Changing the GUI

Various GUI modifications can be selected as seen next.