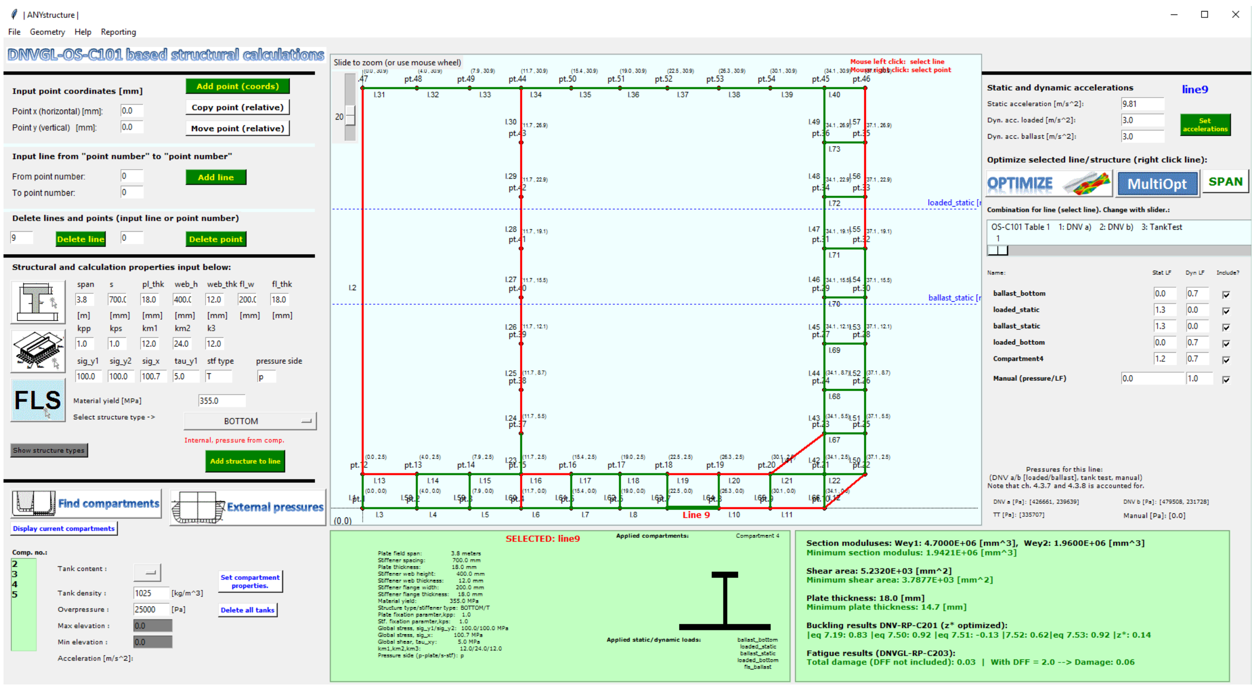
ANYstructure documentation



April 2019

Version 0.6.x

Table of contents

Modelling

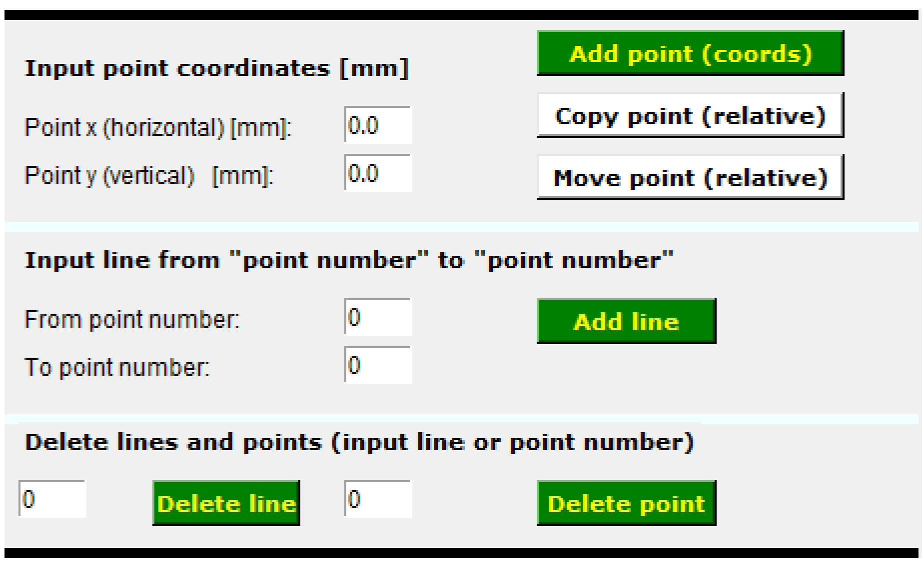
Modelling is done in upper left corner.

Right click: **select point**

You can copy or move the selected point by shortcut or clicking Buttons.

Left click: **select line**

A line is made by right clicking two points (or input point number)



Speed up your modelling significantly by using the shortcuts:

CTRL-Z Undo modelling

CTRL-C Copy a selected point

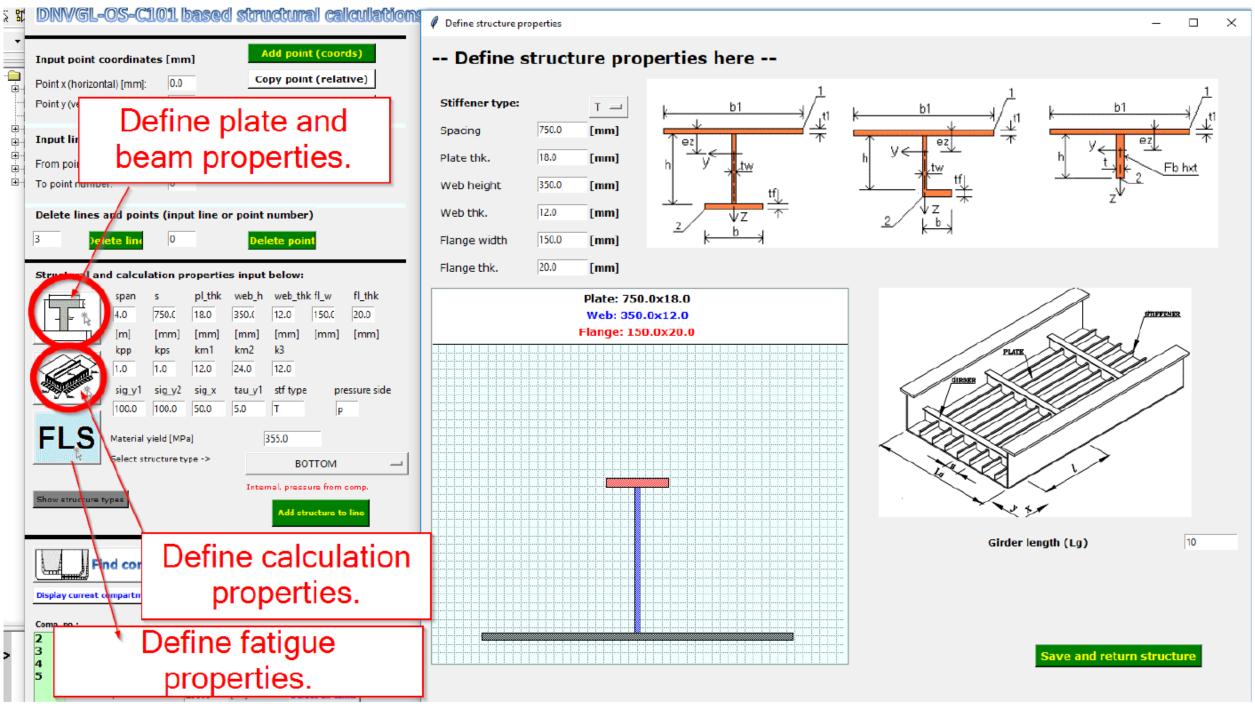
CTRL-M Move a selected point

CTRL-Q New line between two selected points

CTRL-S Assign properties to a selected line

Assigning properties

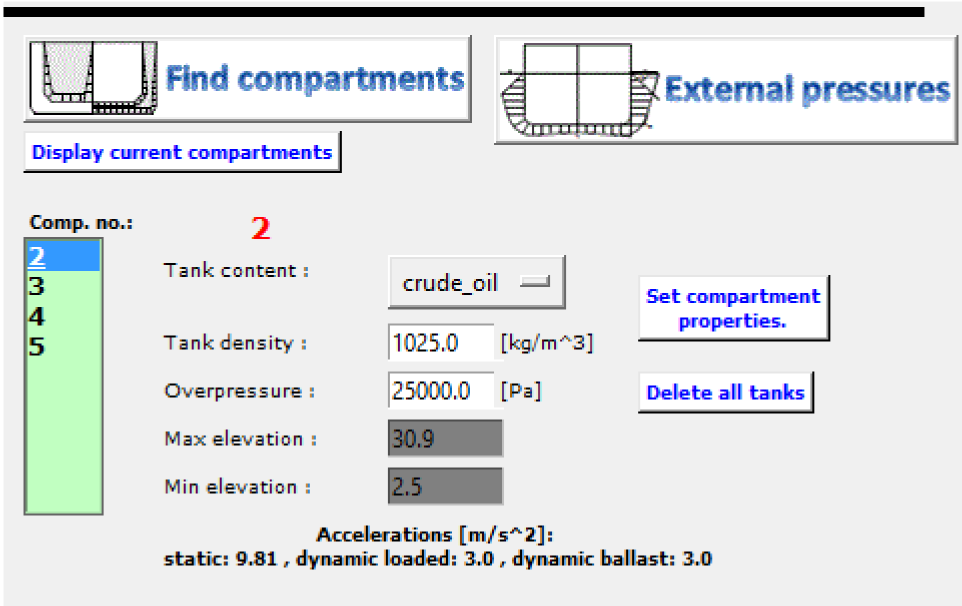
Input properties manually or click the button indicated below to set the values. Values are set by clicking “Add structure to line”. This also applies to fatigue properties.



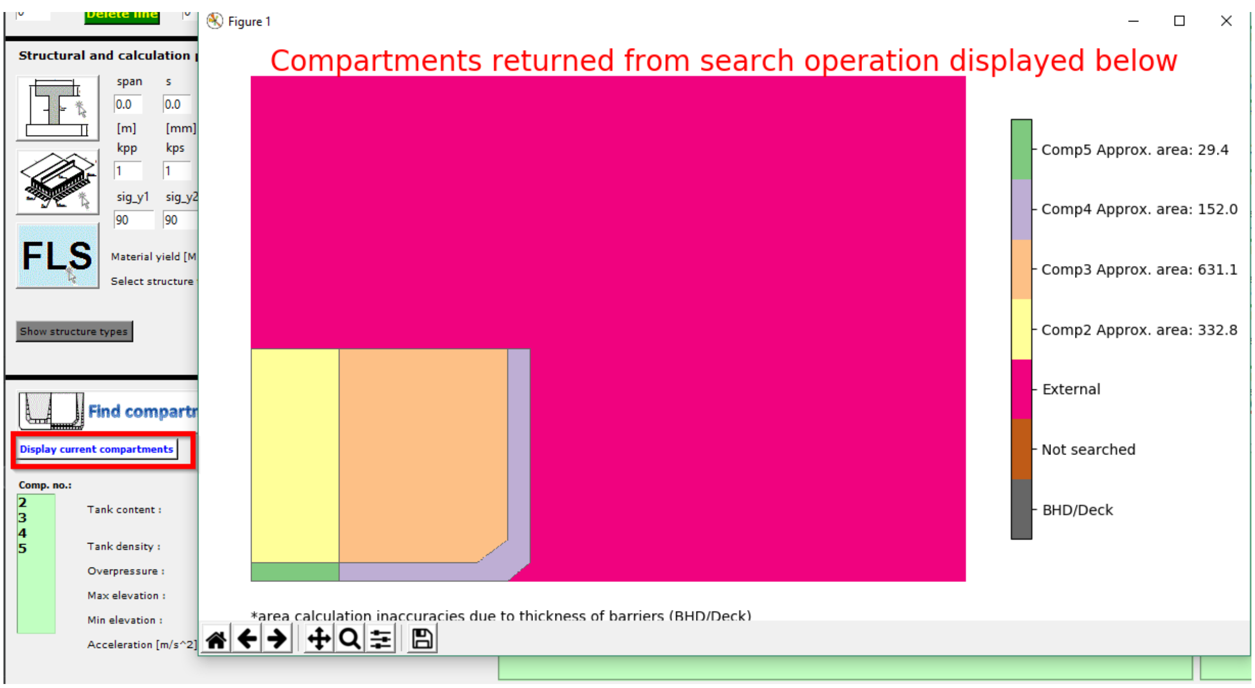
Define tanks

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

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



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



Define external pressures

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

NOTE:

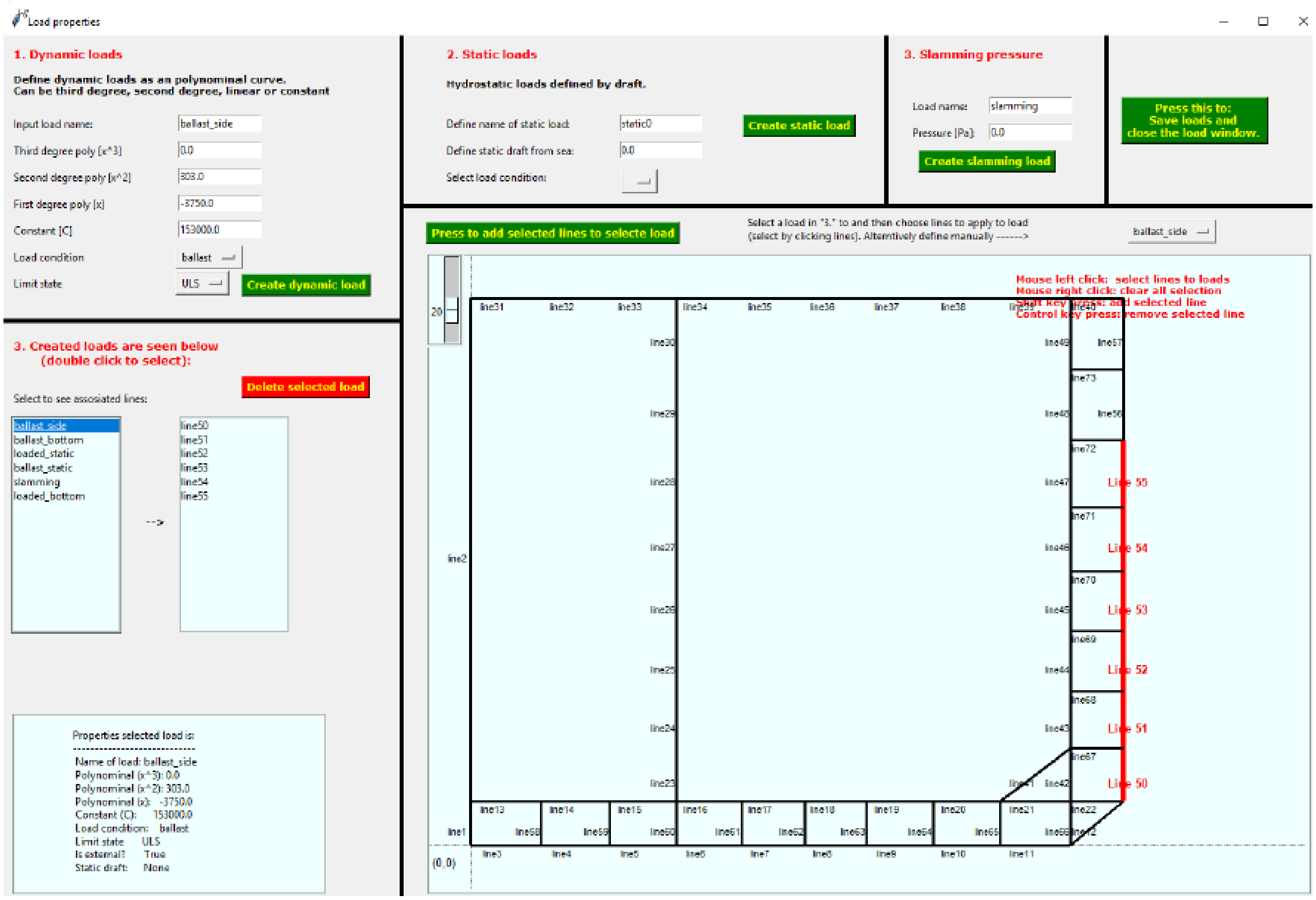
FOR DYNAMIC EQUATION THE FOLLOWING APPLIES

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

Y (vertical) used for BBS, SIDE_SHELL, SSS

After new window is opened:

1. **Make dynamic loads**
   a. **Dynamic loads are made by defining up to 3rd degree equations. X or Y direction depends on the defined structure type.**
   b. **Note that you can define a constant dynamic load by using Constant (Constant (C)) only.**
2. **Static loads are calculated according to depth.**
3. **To apply a defined load to a line or multiple lines:**
   a. **a. Select load by clicking the created load**
4. **Click the lines that shall have the load. Click the button "Press to add selected lines to selected load"**
5. **When finished press the button in the upper right corner.**



Load combinations

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

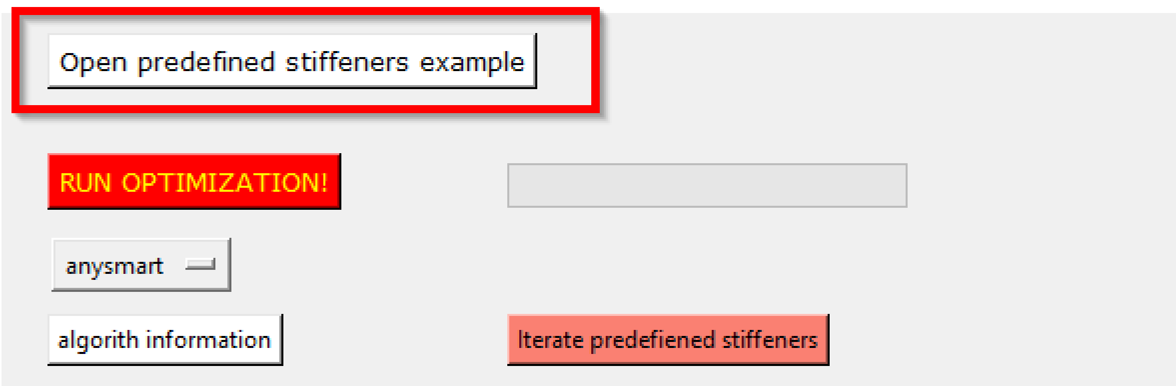
1. **According to DNVGL-OS-C101**
2. **Highest pressure are chosen w.r.t. tank filling.**
3. **You can deselect a load by manually inputting load factor to 0 or deselect include.**

Optimization

Optimization iteration by predefined stiffeners

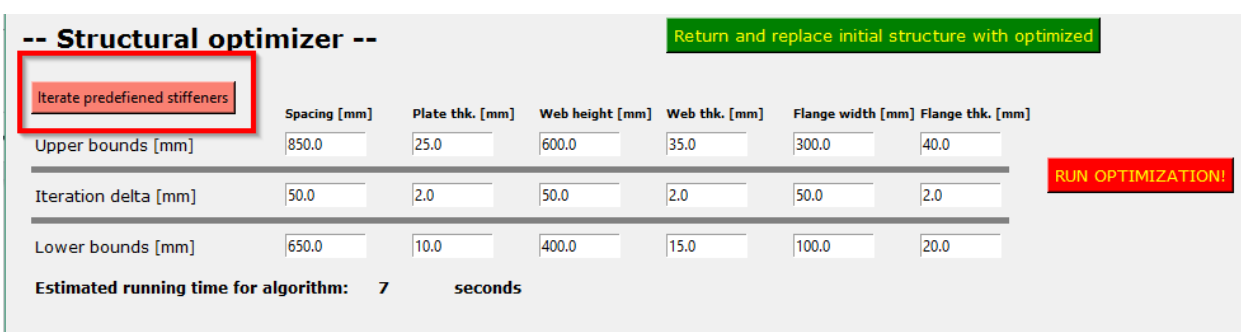
From 0.5 you can iterate by a defined set of stiffeners. Press the button marked below. Open a csv (or json) file. Then start your iterations. The only other input is the stiffener spacing and plate thickness.

To see how the input format is click the "open predefined stiffeners example" button. See illustrations next.



Note that the weight of your initial structure is ignored even though it is calculated. If the initial structure is in your predefined set it will be included in the evaluations.

Press the button indicated below to activate. A open file window will open when running the optimization.

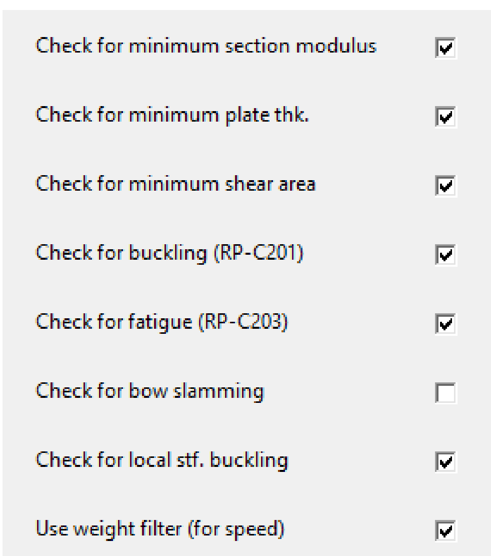


Single optimization

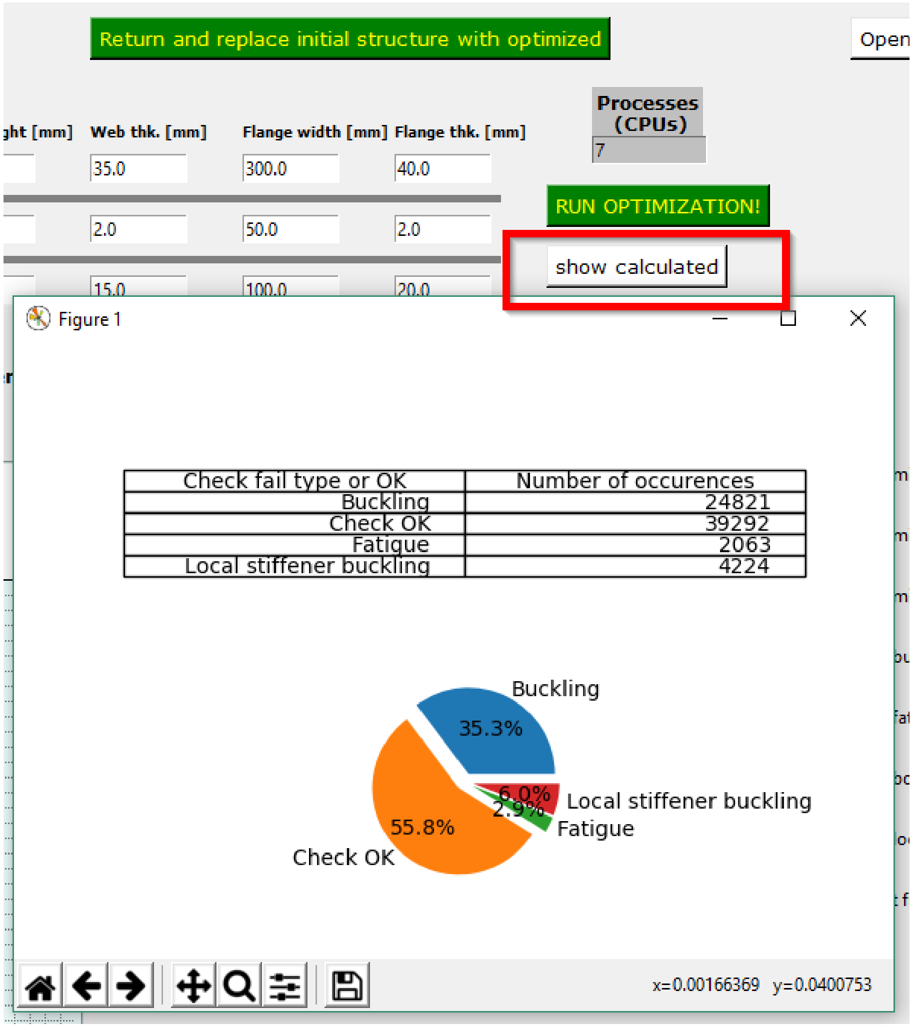
Single optimization is done by clicking a line and clicking the "OPTIMIZE" button.

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

Various checks in the optimization module:
You can select the checks to be performed.
The weight filter ensures that only sections with a lower weight than the current minimum weight. This significantly speed up the calculations, but if you want to see the full distribution of the various checks this must be unchecked.



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



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

Multiple optimization

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

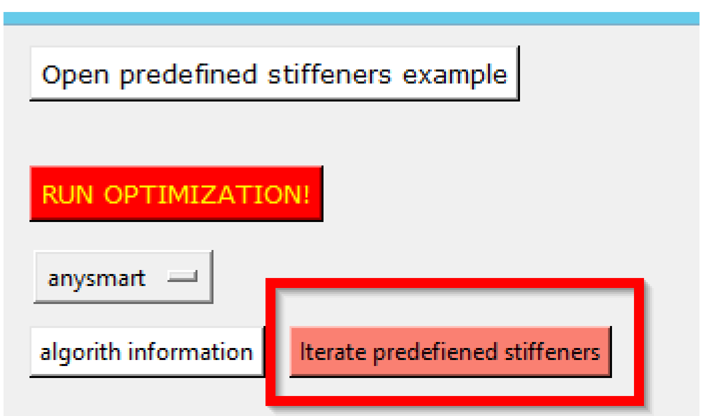
1. **Same input on upper bounds, lower bounds and delta.**
2. **Click all the lines you want to include in the optimization.**
3. **Run the optimization.**
4. **Check the properties by middle clicking the line you ran.**
5. **If you are happy return the properties by clicking the top button**

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

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

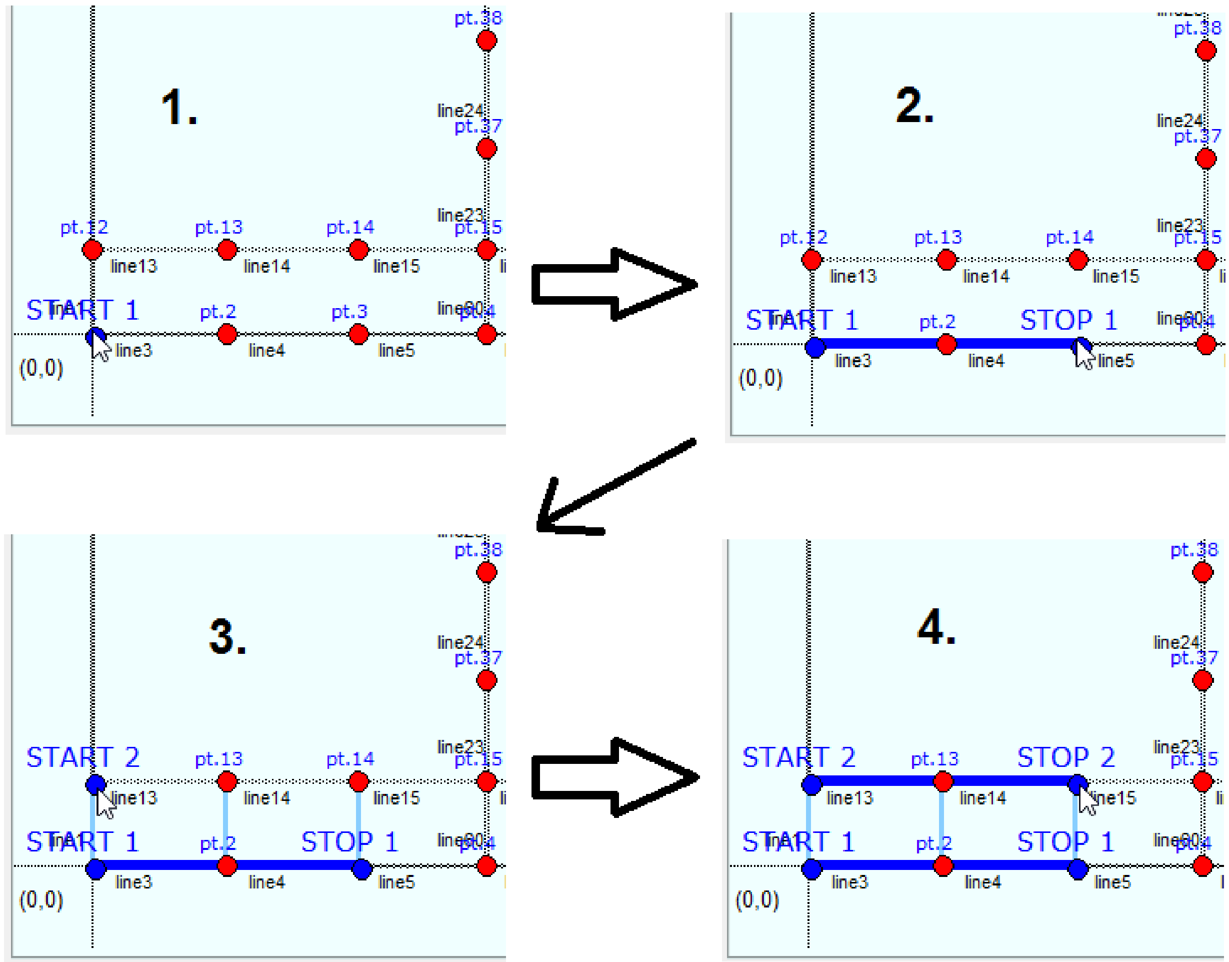
Span optimization

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



The optimization is started as follows.

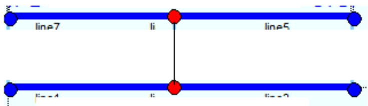
1. **Start by clicking as illustrated next:**



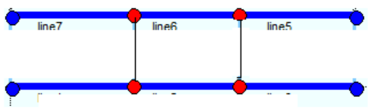
2. Then run optimization.

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.

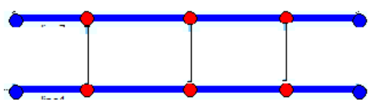
4 plate fields



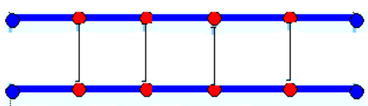
6 plate fields



8 plate fields



10 plate fields

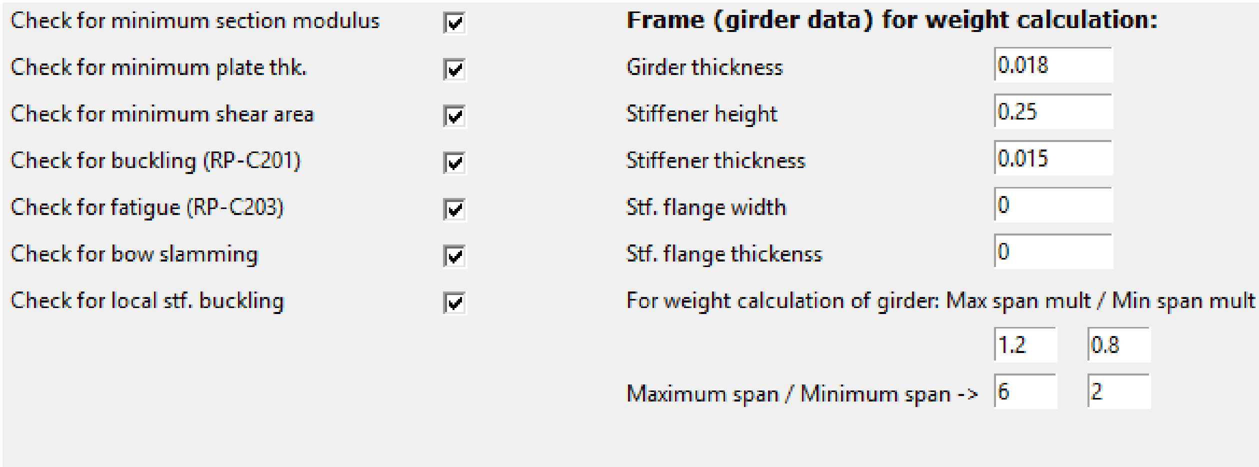


You can, similar to single optimization, select the checks that shall be runned. Also you can set the girder (frame) properties. This is used for calculating the weights.

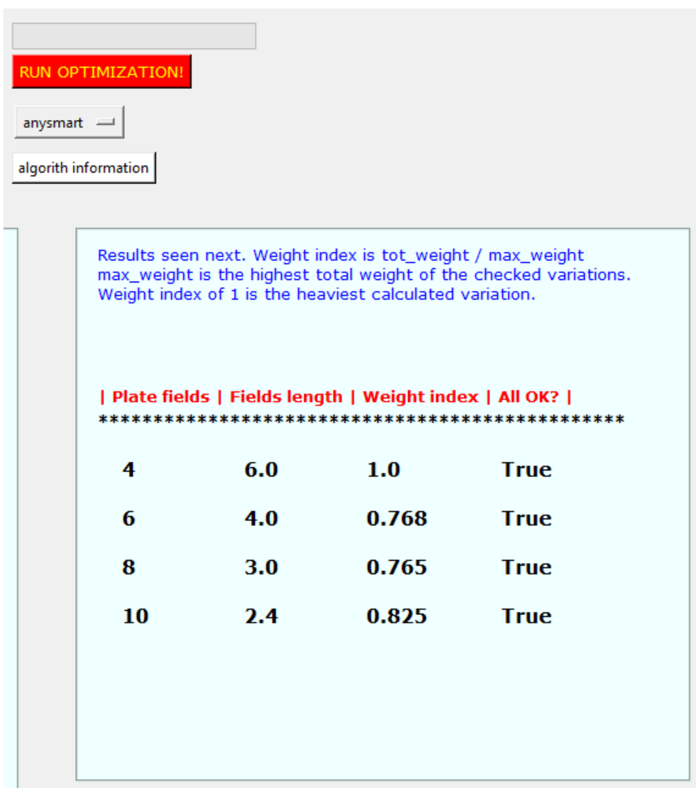
When the analysis has been runned you should save your results. Just specify a file name in the save file dialog.

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

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



Results are presented as seen next.



In this case 8 plate fields with length of 3 meter will give the lowest weight. 6 plate fields is almost equal.

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