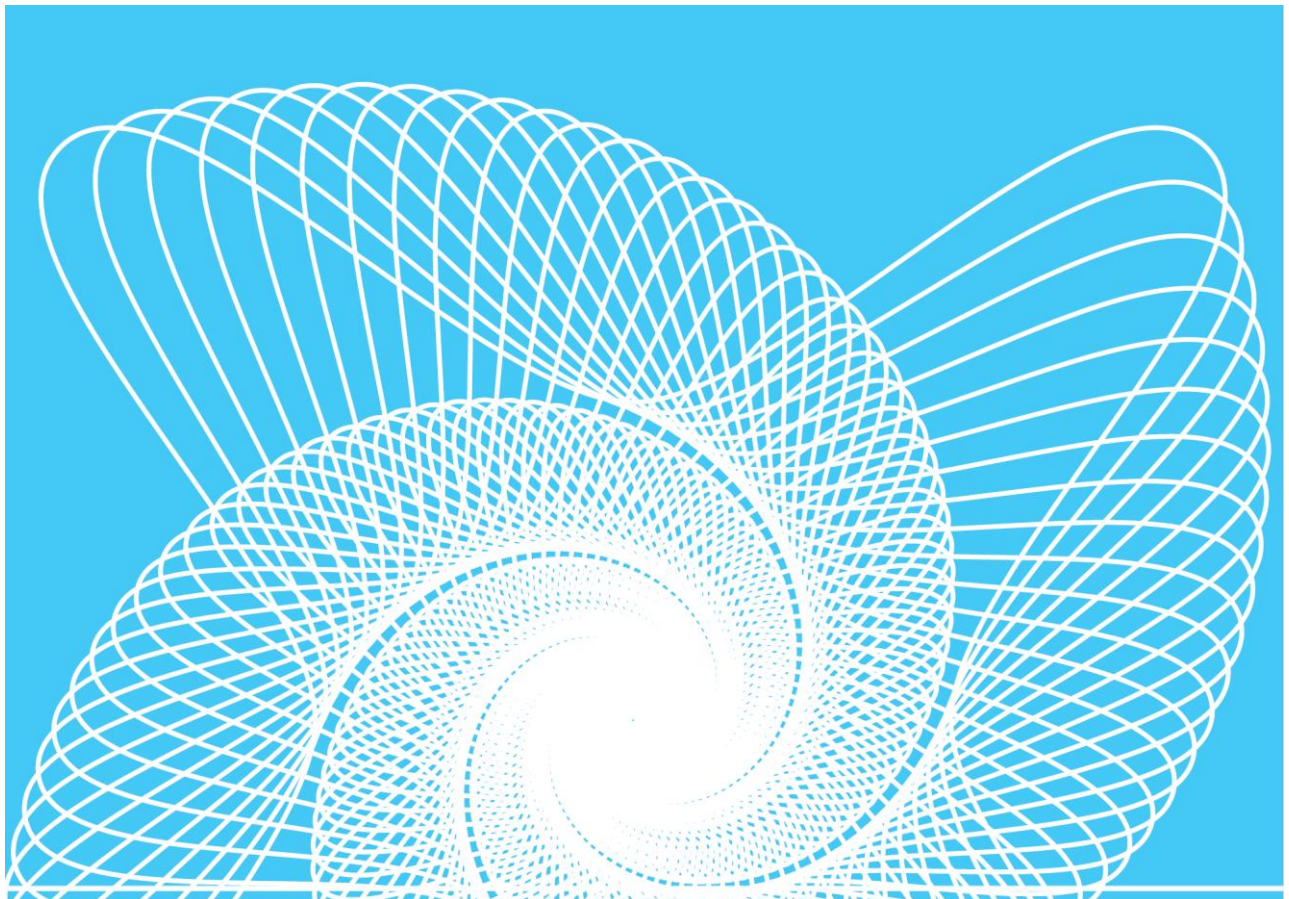


Tutorial

NAUTICUSTM Hull

PSM grillage - modelling and analysis

DNV Rules Pt.3 Ch.6 Sec.6 [2.2]





© DNV AS. All rights reserved

This publication or parts thereof may not be reproduced or transmitted in any form or by any means, including copying or recording, without the prior written consent of DNV AS



Table of contents

1	INTRODUCTION	1
1.1	Limitations	1
2	IMPORT 3D-BEAM MODEL INTO NAUTICUS HULL	2
2.1	Import the workspace	3
2.2	Import 3D-Beam model	4
2.3	Prepare the 3D-Beam model for rule check	6
3	HULL GIRDER STRESSES	10
3.1	Midship cross-section properties	10
3.2	Hull girder properties	12
4	GENERATE LOADS AND RUN	13
4.1	Get Rule loads	13
4.2	Run the analysis	15
5	RULE CHECK RESULTS	16
5.1	Stress Check	16
5.2	Pillar buckling	18

1 INTRODUCTION

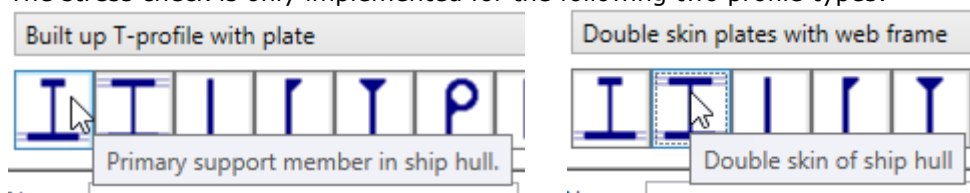
The purpose of this tutorial is to provide the user with a systematic approach to familiarize her-/himself with rule check calculations on PSM grillage structures in Nauticus Hull version 20.0 and newer, on a general level.

Primary Structural Members are modelled in 3D-Beam for the area that is to be analysed. The PSM grillage tool will add rule loads according to the DNV rules to this 3D-Beam model, and run a static strength analysis in 3D-Beam. Analyses results are checked according to rule requirements. The rule check includes

1. Stress check
2. Pillar buckling check

1.1 Limitations

- The PSM grillage tool is only available for DNV rules.
- The stress check is only implemented for the following two profile types:



- There is no limitation with regard to profile type in the pillar buckling check, but results for other profiles than those listed in DNVGL-CG-0128 sec.3 [4.4] Table 8, should be evaluated carefully.
- Only two loading conditions, i.e. one ballast and one full-load condition, is supported. If several are specified, results will be overwritten by the **last loading condition**.

For instance, if there are multiple Homogenous full load conditions, it is the last condition defined in Compartments and Loads that will be used.

2 IMPORT 3D-BEAM MODEL INTO NAUTICUS HULL

First, extract "Tutorial PSM grillage model files" file by clicking on it:

Tutorials and example models

Latest updates of Nauticus Hull tutorials and example models are available in [Customer portal](#). (Login required)


Nauticus Hull
v20 tutorial


Nauticus Hull v20
tutorial model files


Tutorial
PSM grillage


Tutorial PSM
grillage model files


Update compartment
data in Nauticus Hull

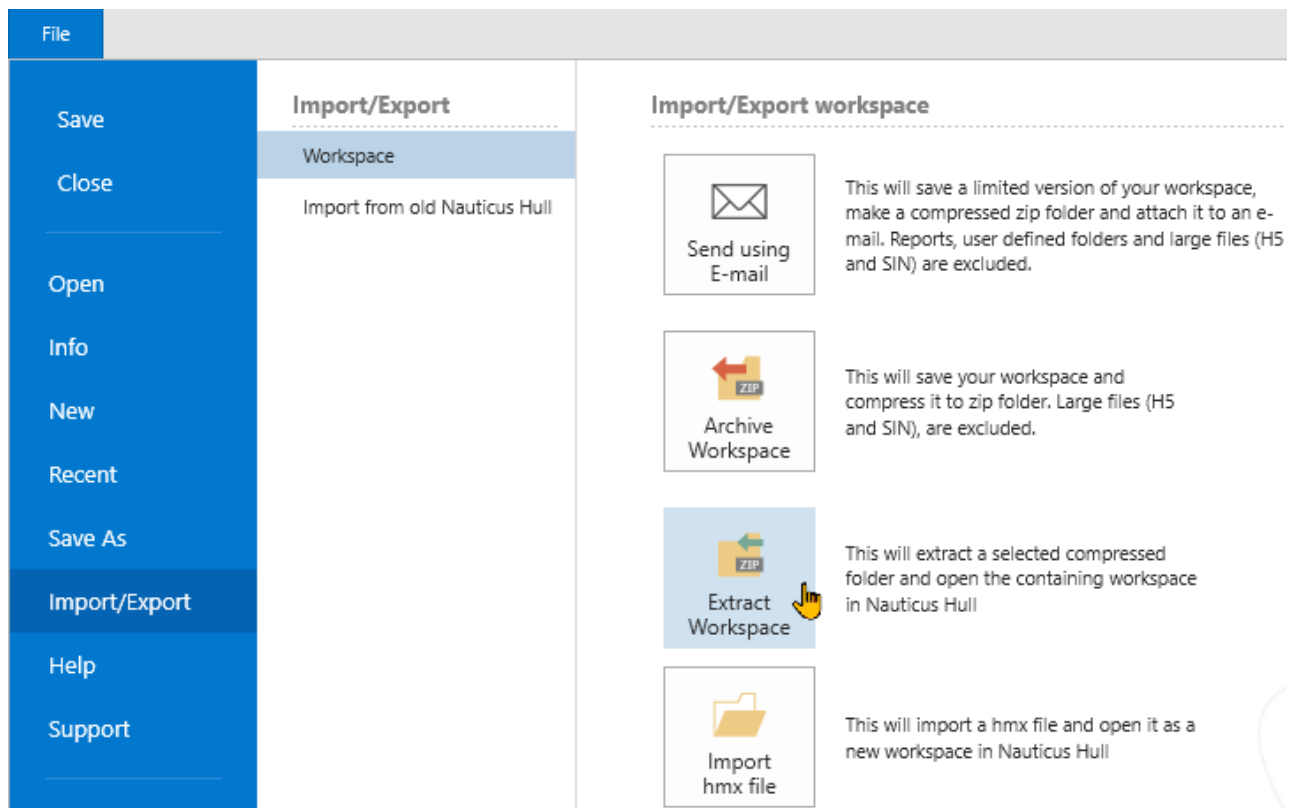
Rules and standards

Extract to a location on your disk, for instance C:\Documents\DNV.

System (C:) > Documents > DNV > NauticusHullv20_Tutorial_PSMgrillage_model_files >				
	Name	Date modified	Type	Size
	 OSV PSM.clb	08.04.2024 14:22	CLB File	1 464 KB
	 OSV PSM.OSV PSM.zip	08.04.2024 14:33	ZIP archive	1 491 KB

2.1 Import the workspace

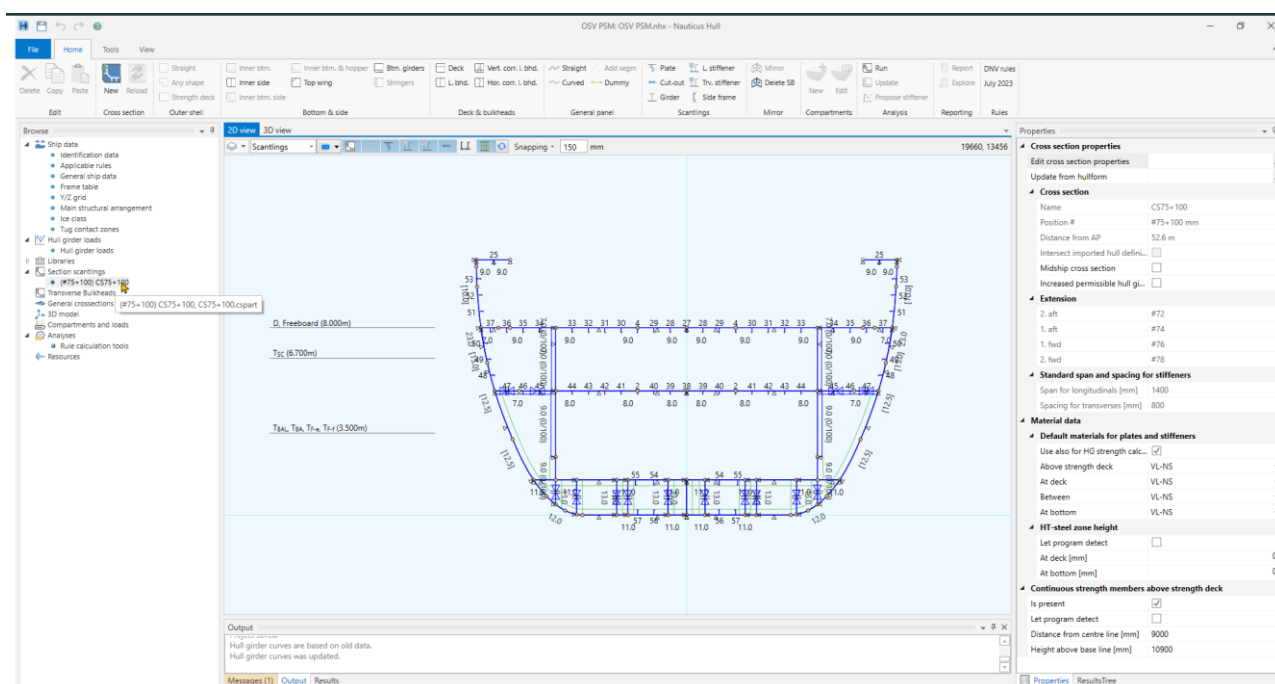
- Start Nauticus Hull
- Select File | Import/Export
- Select Extract workspace



Navigate down to the folder where you extracted the files and select **OSV PSM.OSV PSM.zip**

> System (C:) > Documents > DNV > NauticusHullv20_Tutorial_PSMgrillage_model_files		
Name	Date modified	Type
 OSV PSM.OSV PSM.zip	08.04.2024 14:33	ZIP archive

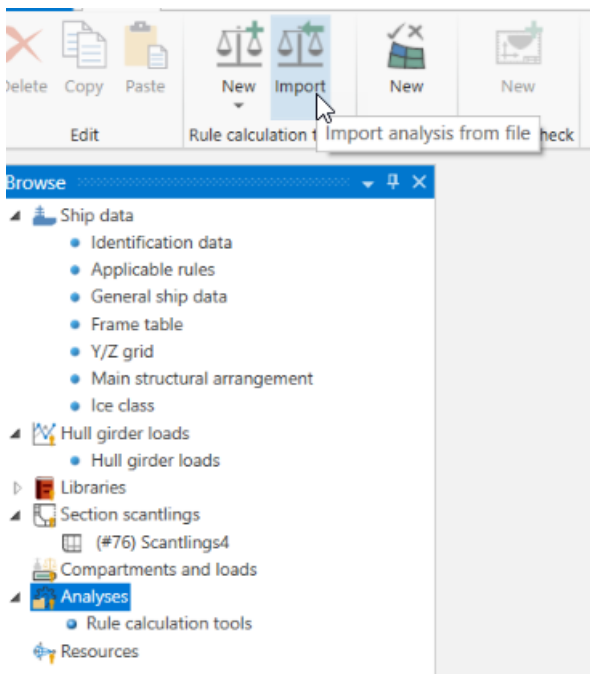
This workspace contains a cross-section model of the machinery area of an OSV. Our grillage model will be located in the same area as the cross-section:



Familiarize yourself with the vessel's ship data, applied rules, compartments and loads etc.

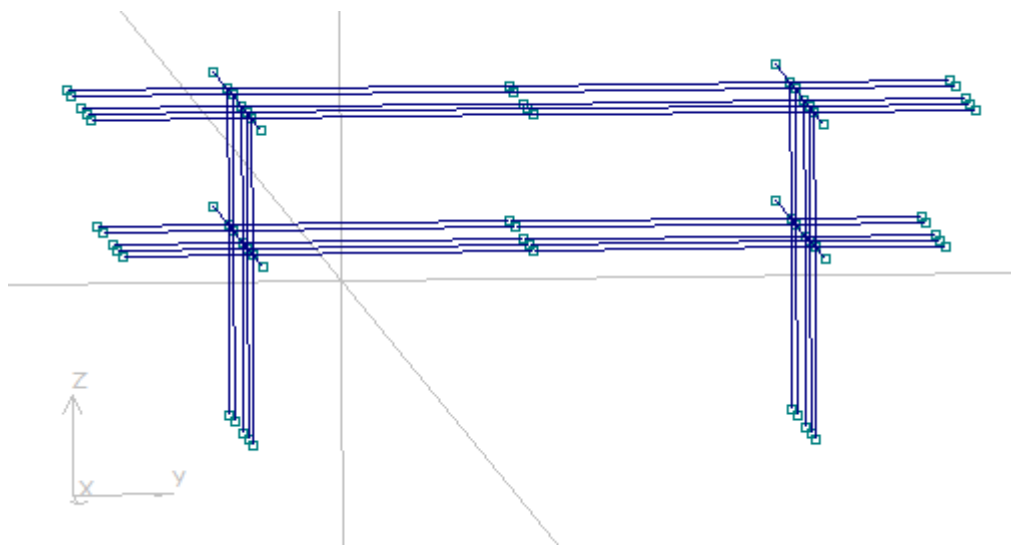
2.2 Import 3D-Beam model

Next step is to import a predefined 3D Beam model into workspace:



Select file **OSV PSM.clb** from the folder that you extracted above.

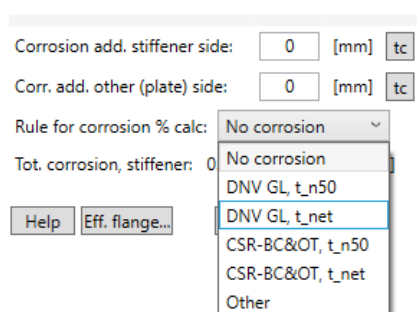
A frame model of the PSM grillage has been prepared in 3D-Beam in advance.



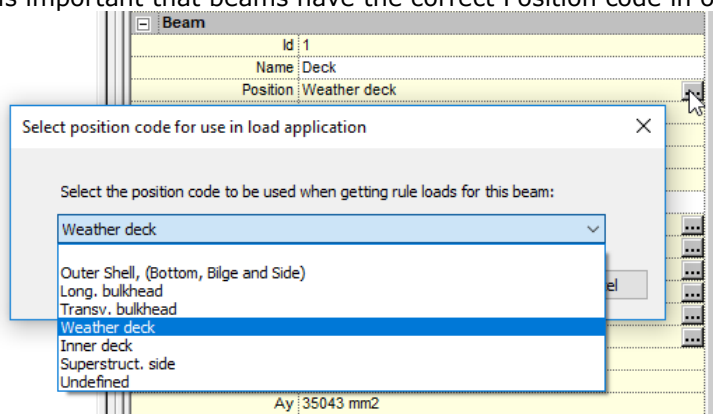
The model consists of geometry and applied properties for each beam element, including the effective breadth of the deck flange. Note that this is a simple model made for illustration only and does **not** follow recommendations for modelling of hull structures represented by 3D beam models.

Hints on modelling:

- It is important that x-position of beams in the 3D Beam model corresponds with the same x-position in Nauticus Hull geometry in order to get correct tank loads applied to the beam elements.
- NOTE: Some guidelines for modelling methods and techniques for beam analyses of ship structures are given in DNV Class Guidelines, DNVGL-CG-0127 Sec. 5.
- Normally gross thicknesses will be specified when modelling in 3D-Beam, and then corrosion addition is calculated according to rules being specified in the profile properties dialogue.

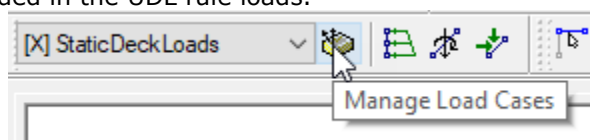


- It is important that beams have the correct Position code in order to get the correct rule loads.

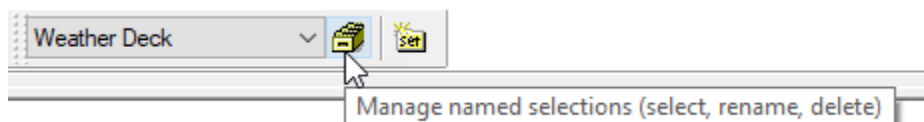


If no position code is specified, the beam will still be a part of the rule check, but rule loads will **not** be applied on the beam.

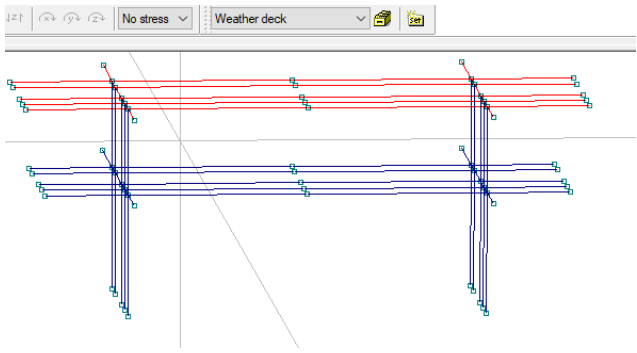
- Static loads shall be applied as an input in order to obtain rule dynamic deck loads. Static uniform deck loads must be applied as beam line loads taking into account the stiffener (or girder) spacing. This loading condition must be named "**StaticDeckLoads**" in order to be included in the UDL rule loads.

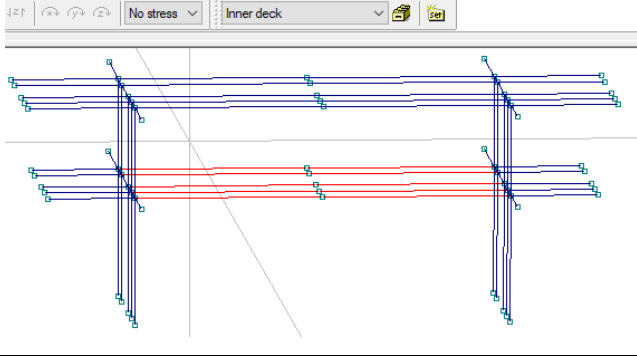
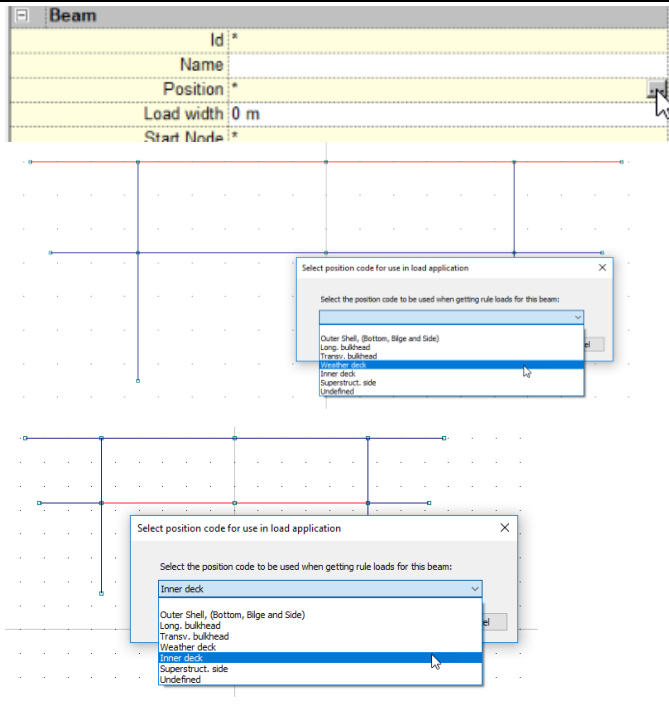


- It may be convenient, especially for large models, to identify parts of the structure with a name, e.g. "Outer shell", "Main supports", "Weather Deck". This allows quick access to these parts for input specification and result analyses:

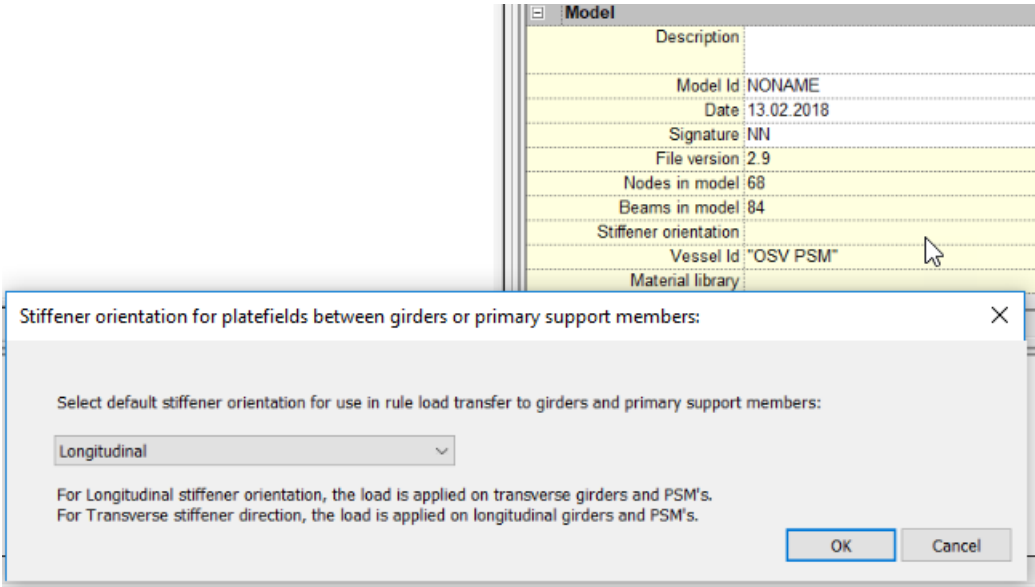


2.3 Prepare the 3D-Beam model for rule check

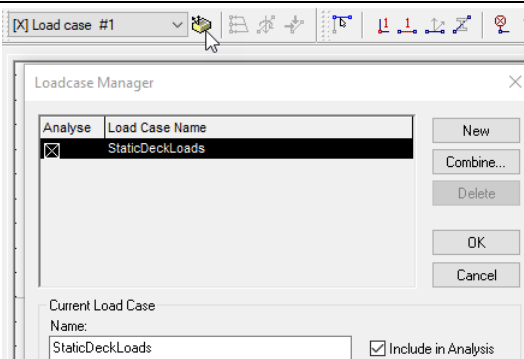
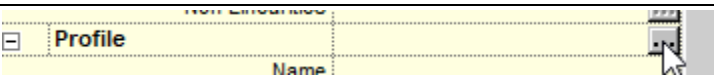
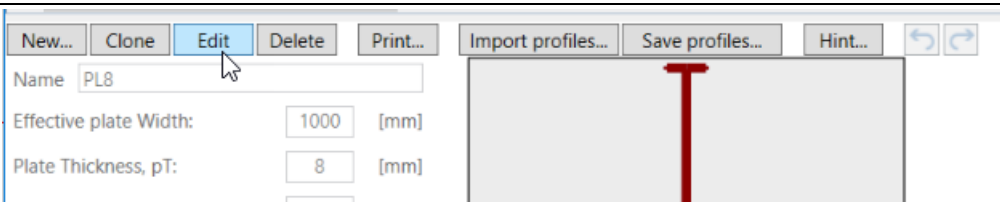
Define all beams on upper level in set 'Weather deck', and all transverse beams at inner lower area (machinery area) in set 'Inner deck'	
---	--

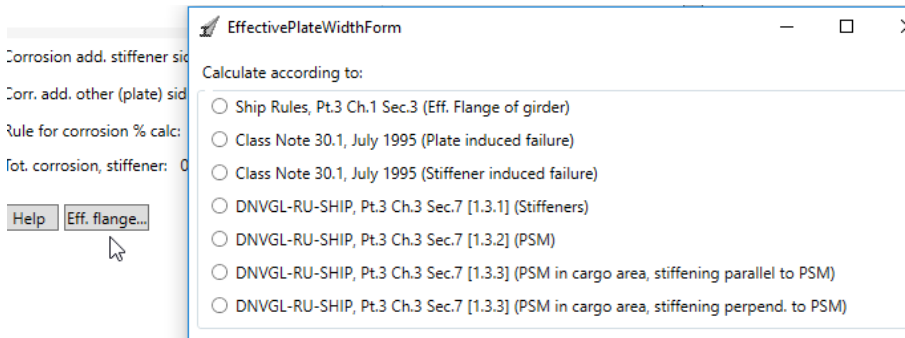
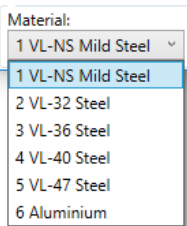
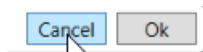
	
Set corresponding Position codes	

No position code is given to the remaining beams, and these will then not receive any rule loads.

Set Stiffener orientation in Longitudinal direction	
--	--

If "Longitudinal" is chosen as stiffener orientation, loads are applied to all transverse beams. If "Transverse" is chosen, loads are applied to all longitudinal beams. Angle tolerance is 45 degrees.

Rename 'Load case #1' to 'StaticDeckLoads'	
Open Profile properties	
Select a profile and Edit	
Press the [tc] - button for more information about corrosion addition.	

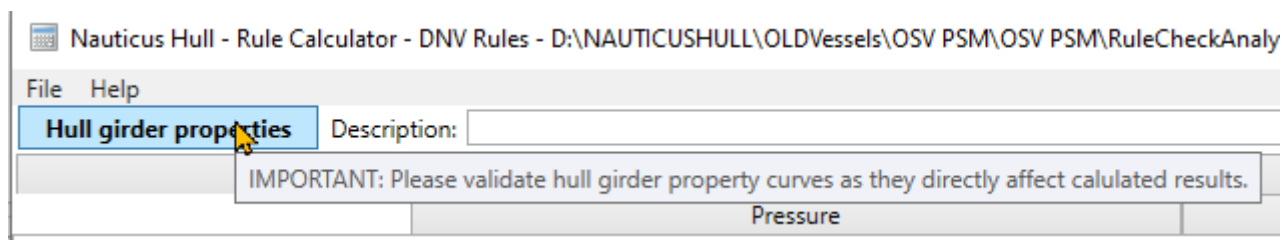
	Corrosion add. stiffener side: 0 [mm] tc Corr. add. other (plate) side: 0 [mm] tc Rule for corrosion % calc: No corrosion No corrosion addition is specified in this model.
Press the [Eff.flange] button	There are several rule references to formulae that can be selected and used to calculate effective flange: 
The model will have material VL-NS Mild Steel	
Press 'Cancel' to close the profile windows	

3 HULL GIRDER STRESSES

Hull girder stresses can be given as input or estimated.

3D Beam has no knowledge of the cross-sections along vessel, and specified hull girder properties will be used together with the rule loads for each loadcase to estimate the hull girder stresses.

To describe the hull girder properties, open an instance of Rule Calculator and access **Hull girder properties**:



Note that this may already have been done as part of other Rule Calculator tasks.

3.1 Midship cross-section properties

First time the Rule Calculator is started a dialogue appears requesting input for midship data describing simplified curves over the whole ship length (ref. 3.2).

These curves will be used as basis for calculating estimated hull girder cross-section properties where this is used in the calculation, e.g. to estimate hull girder stresses.

Hull Girder Properties for Rule Calculator

Name	Unit	Symbol	AP	0.1L	0.2L
Neutral axis	m	Zn	4	4	4
Moment of inertia about vertical neutral axis	m ⁴	Iz n50	9,13	9,13	13,7
Moment of inertia about horizontal neutral axis	m ⁴	Iy n50	2,91	2,91	4,3
Breadth at WL	m	Bx	8,5	15,3	17
Inner Bottom Height	m	IBh	0	0	0
Section modulus, bottom	m ³	Z B-n50	0,73	0,73	1,05
Section modulus, strength deck	m ³	Z D-n50	0,73	0,73	1,05
Section modulus, top deck	m ³	Z Top-n50	0,73	0,73	1,05
Minimum moment of inertia about horizontal axis	m ⁴	IyR	0	0	0
Minimum net section modulus	m ³	Z R	0,55	0,55	0,82

Information from existing cross sections

Name	Zn	Iz	Iy	IBh	Bx
☐ (#75) cs75tst	4	18,26	5,82	0	17
☒ (#75+100) CS75+100	4,25	33,24	12,43	1,5	17,14

Input midship data

In order to do calculation following data needs to be defined. Please enter data for midship.

IyR Minimum moment of inertia about horizontal axis:	0,0	m ⁴
ZR Minimum net section modulus:	1,091	m ³
Zn Neutral axis:	4	m
Iz n50 Moment of inertia about vertical neutral axis:	18,26	m ⁴
Iy n50 Moment of inertia about horizontal neutral axis:	5,82	m ⁴
IBh Inner Bottom Height:	1,5	m
Z0 Height of strength deck at side: *	8	m
ZvD Height of equivalent deck line: *	18	m

* Z0 and ZvD are optional input used for making section modulus curves. Only used for information.

Information from existing cross sections

Double click on a row to use data from that cross section.

Name	Pos	Zn	Iz	Iy	IBh
(#75) cs75tst	#75	4	18,26	5,82	0
(#75+100) CS75+100	#75+100 mm	4,25	33,24	12,43	1,5

Input midship data

×

In order to do calculation following data needs to be defined. Please enter data for midship.

I_{yR} Minimum moment of inertia about horizontal axis:

254.4

m^4

Z_R Minimum net section modulus:

32.897

m^3

Z_n Neutral axis:

0

m

$I_{z_{n50}}$ Moment of inertia about vertical neutral axis:

0

m^4

$I_{y_{n50}}$ Moment of inertia about horizontal neutral axis:

0

m^4

IB_h Inner Bottom Height:

0

m

Z_D Height of strength deck at side: *

21

m

Z_{VD} Height of equivalent deck line: *

21

m

* Z_D and Z_{VD} are optional input used for making section modulus curves. Only used for information.

OK

Cancel

These values for the midship section apply to the whole length of the vessel if not defined otherwise. To modify values for area beyond the midship, open the Hull Girder Properties window from Common data or the button on the upper right side.

If there are cross sections models available, calculated data from these will be shown at the lower part.

Data from the cross section with lowest I_z will be proposed as default values. To use data from another cross section select the row, right-click and select menu 'Use data from selected row as midship data' or just double click on row. Please note that this table of information will not be shown if no cross section models have been calculated.

Information from existing cross sections

Double click on a row to use data from that cross section.

Name	Pos	Z_n
(#10) Aft Body FR15	#10	18,11
(#25) Engine room_AFT FR18	#25	17,19
(#35) Engine room_M_AFT FR37	#35	14,09
(#41) Engine room_M_AFT FR39	#41	16,71
(#46) Engine room_M FR45	#46	16,54
(#54) Engine room_M FR52	#54	16,19
(#60+425) Engine room_F FR59	#60+425 mm	15,8
(#63+2555) No5 Cargo Hold FR62 TBD	#63+2555 mm	14,99
(#65+2555) No5 Cargo Hold FR65 S.FWD	#65+2555 mm	14,18

The input data definition for DNV HSLC Rules is as shown:

Input midship data

×

In order to do calculation following data needs to be defined. Please enter data for midship.

Z_n Neutral axis:

0

m

I_y Moment of inertia about horizontal neutral axis:

0

m^4

Z_D Height of strength deck at side: *

5,25

m

* Z_D is optional input used for making section modulus curves. Only used for information.

OK

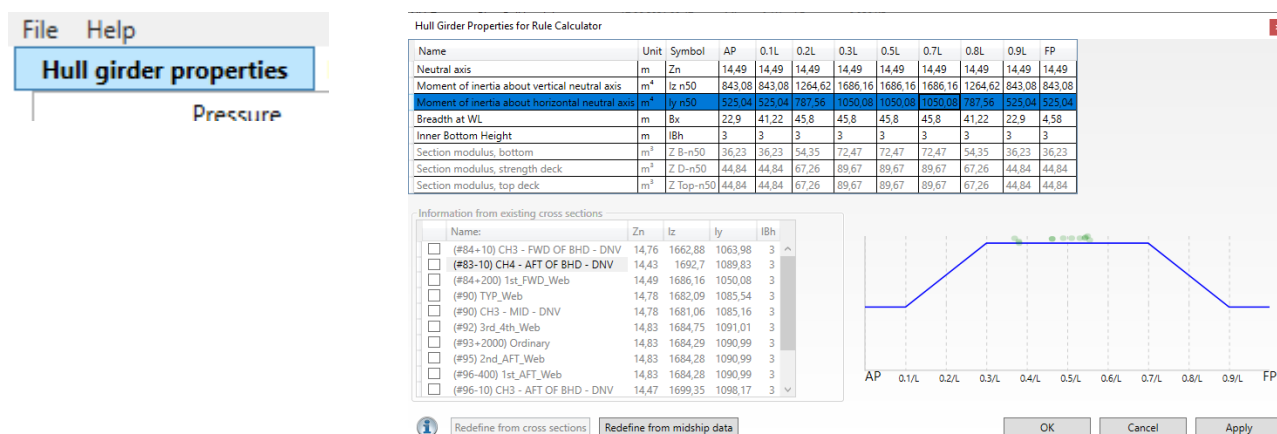
Cancel

3.2 Hull girder properties

The Hull Girder Properties window shows simplified cross sectional data along the entire ship length and the distribution curves are normally made from midship data. The hull girder property data will be used in all calculations, including hull girder bending stress, σ_{hg} . Data from these curves are used even if a cross section model is available at calculated position.

The distribution curves are pre-defined by the program and are representing assumed cross sectional data along the ship length. Please note that these curves are not defined by the Rules and should be adjusted by the user if found necessary. The assumed moment of inertia at AP and FP is 50% reduced compared to amidships. The distribution curves are not ship type dependent.

Some values in the data tables are editable. It is also possible to add new columns to the data table if needed.



One can always redefine curves by pressing the button 'Redefine from midship data'.

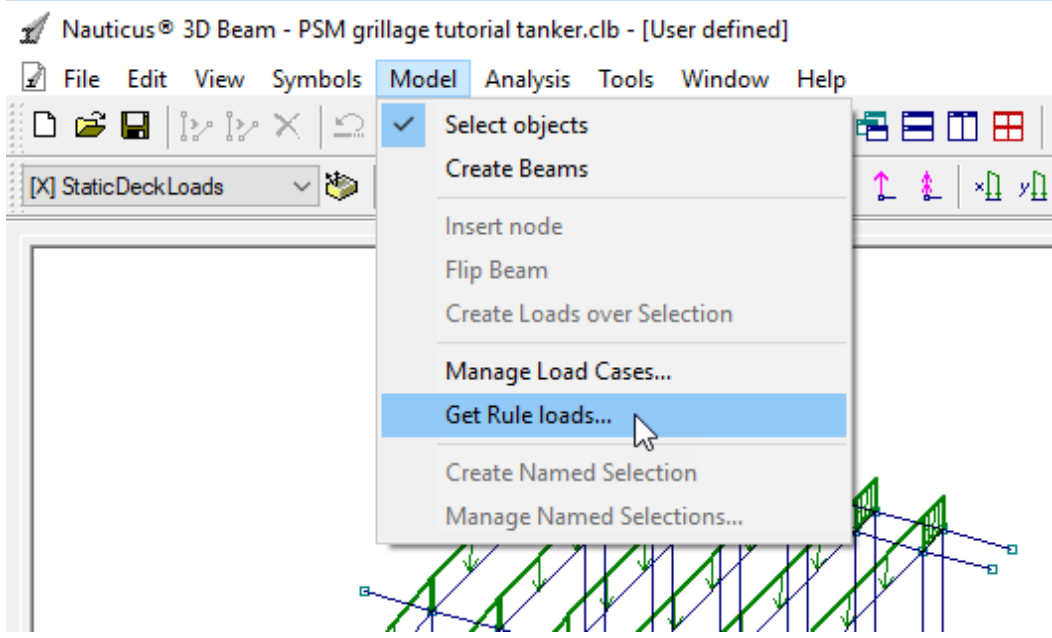
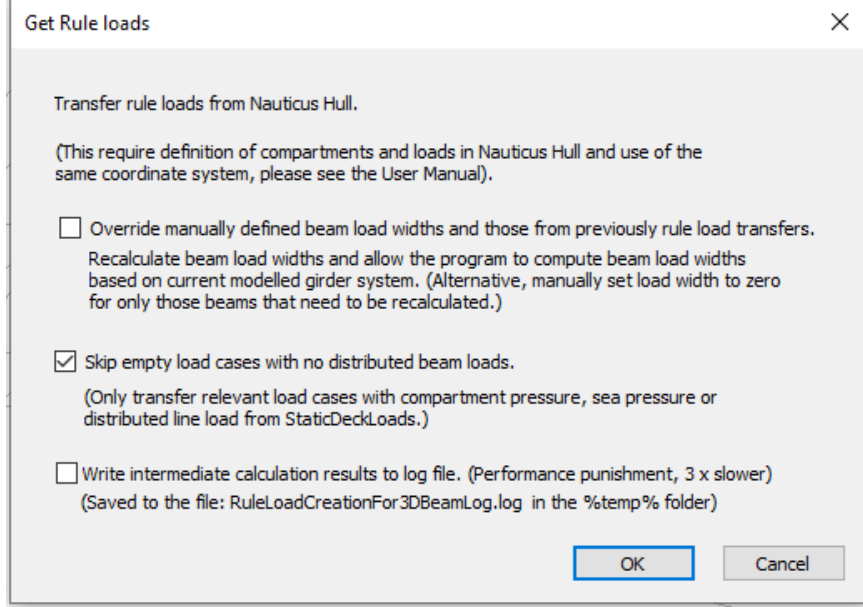
Data from existing cross section models may be shown as thick green lines or dots in the graph. This is just for reference, but may be used for verification of correct input. When there are data from several calculated cross section available, user might select cross sections by check-box and press button 'Redefine from cross sections'.

User should always verify curves after generation and edit if found necessary. Additional positions/columns may be added by right click and select 'Insert new column'.

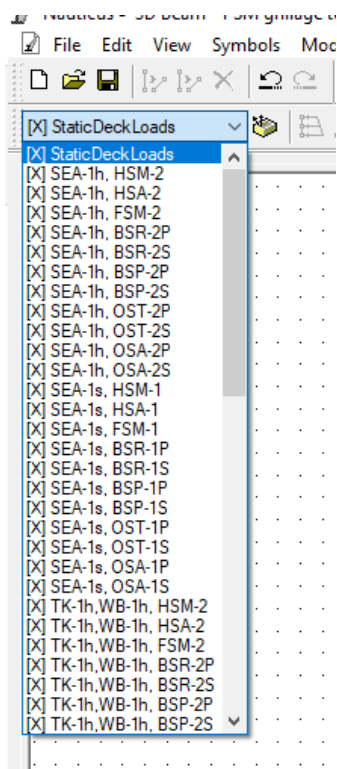
The hull girder property data will be stored in the ShipData.xml. This information will be common and retrievable every time a new Rule Calculator file is created.

4 GENERATE LOADS AND RUN

4.1 Get Rule loads

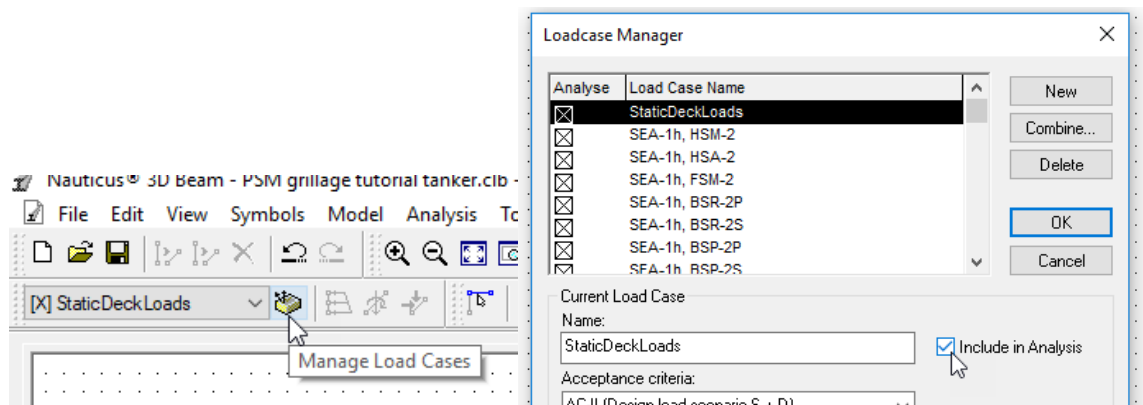
Model → Get Rule loads	
	There are no previously defined beam load widths, so it is not necessary to tick "Override"  Empty load cases can be skipped.

The rule load cases will be added and by default included in the calculations:

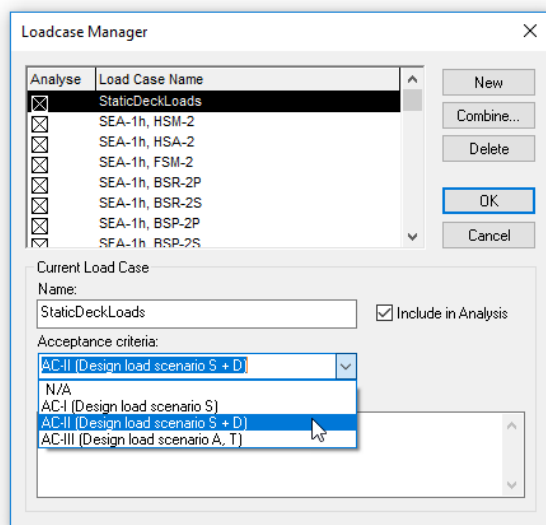


If desired, it is possible to use the Loadcase manager to

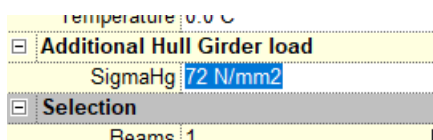
- exclude cases



- to make new cases or combinations,
- or to change rule acceptance criteria:

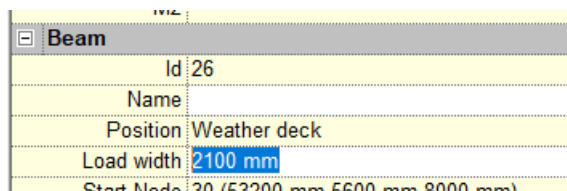


Notice that longitudinal beams now, for each new loadcase, also have received additional Hull Girder stresses



These stresses can be deleted if desired, e.g. in superstructures.

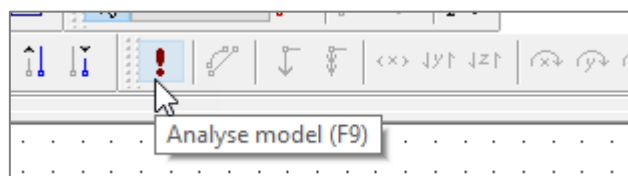
Also notice that the load width that is used in calculation of line loads for the rule loads now is displayed:



If you want to use a different width than calculated by the program, it is possible to update this value and then run the "Get Rule loads..." – command once more.

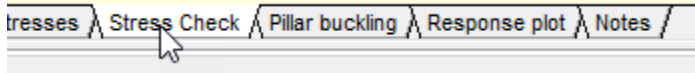
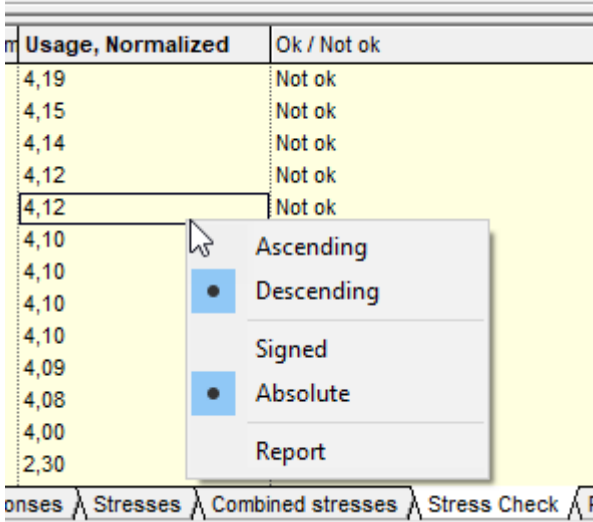
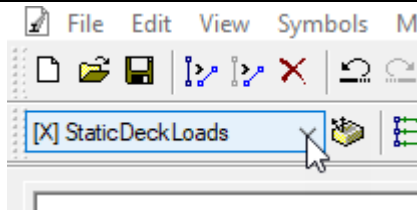
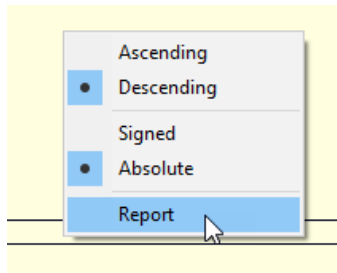
4.2 Run the analysis

Push analysis button to run the rule check:



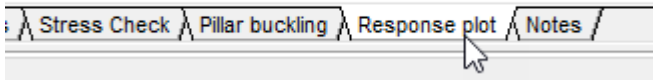
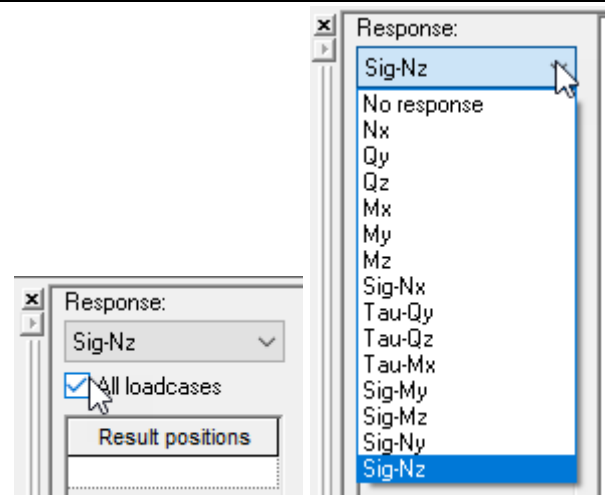
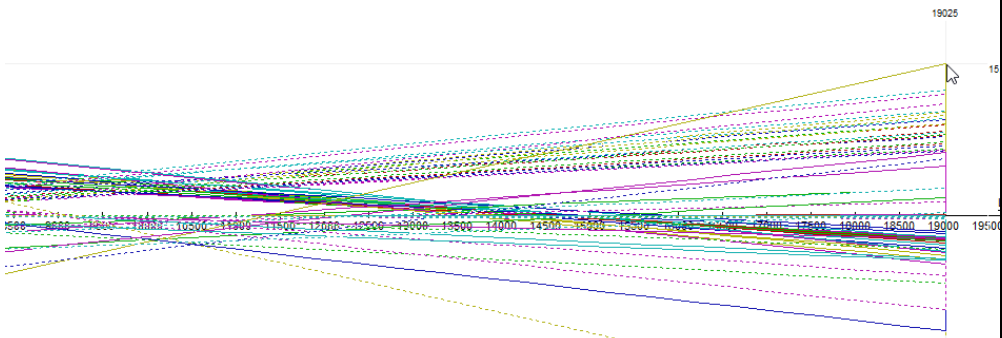
5 RULE CHECK RESULTS

5.1 Stress Check

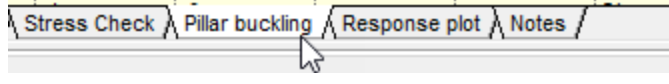
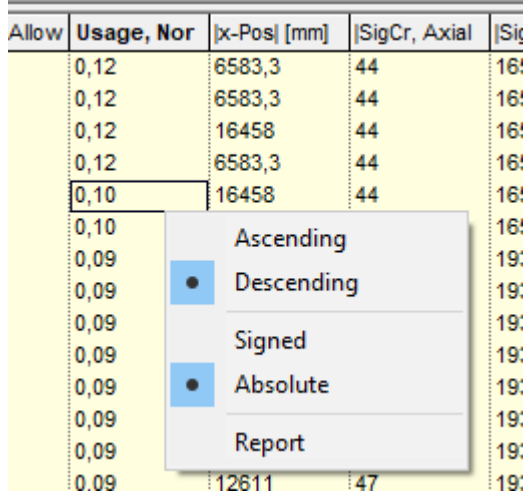
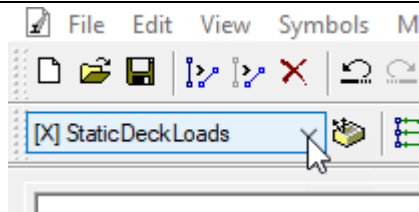
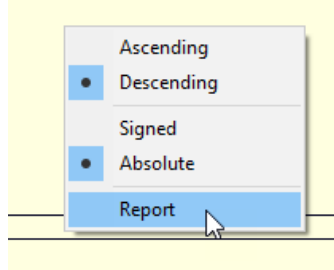
Select tab 'Stress Check'	
Double-click in the "Usage, Normalized" column to sort on highest usage factor. Right-click and select Descending and Absolute	
Select load condition StaticDeckLoads	
Press the "Down-arrow" on the keyboard	This is a convenient way to loop through all loading conditions to check max. Usage factors.
Right-click in the output pane and select Report	
Go to the bottom of the report	Here you will find some rule references and explanations:

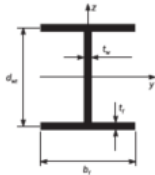
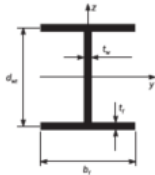
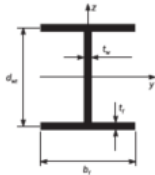
	Abbreviations Verification of beam stresses as specified in Rules for classification: Ships – DNVGL-RU-SHIP Pt.3 Ch.6. [2.2.2], Edition Jan 2018 and later. Direct strength - beam analysis, Acceptance criteria for beam analysis. Stress check information: PSM: Primary support member, Beam identification number Name: User's beam identification Hull girder stress: Average axial stress in member from global bending of hull girder, σ_{bg}, calculated by rule calculator. Bend. stress: Worst Bending stress along the beam from local loads and beam node rotations. Allow. Bend: Allowed bending stress = $C_b \cdot R_{bh}$. Shear stress: Worst average shear stress in the web-plate along the beam. Allow. Shear: Allowed shear stress = $C_s \cdot R_{sh}$. Normalized usage: The maximum usage factor, calculated as stress divided with the allowable stress. A value < 1.0 means that the PSM is ok. Is Ok: Text showing Ok or Not ok.
--	---

All loadcases can be displayed simultaneously for one force or stress component for one beam:

Select tab 'Response Plot'	
Tick on "All loadcases" and select one component in the dropdown menu	 Max values are shown on the right end when cursor hovers on the curves. 

5.2 Pillar buckling

Select tab 'Pillar buckling'	
Double-click in the "Usage, Normalized" column to sort on highest usage factor. Right-click and select Descending and Absolute	
Select load condition StaticDeckLoads	
Press the "Down-arrow" on the keyboard	This is a convenient way to loop through all loading conditions to check max. Usage factors.
Right-click in the output pane and select Report	
Go to the bottom of the report	Here you will find some rule references and explanations:

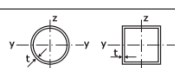
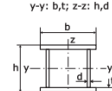
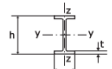
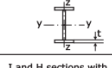
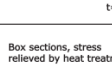
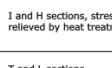
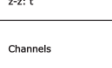
	Pillar properties: No.: Beam identification number Length: Elastic length, (total length minus rigid ends), of beam. f_{end-y}: End constrain factor of the pillar, for bending about y-axis of beam. f_{end-z}: End constrain factor of the pillar, for bending about z-axis of beam. (see DNVGL-RU-CG-0128, sec.3 [4.2] Table 7) <table><tr><td></td><td></td><td></td><td></td><td></td><td></td><td></td></tr><tr><td>f_{end}</td><td>4.0</td><td>2.0</td><td>1.0</td><td>1.0</td><td>0.25</td><td>0.25</td></tr></table> Ax: Axial area (total profile area) Ix: Torsional moment of inertia Iy: Moment of inertia about local y-axis Iz: Moment of inertia about local z-axis Braced joint transl. Yes, if both beam ends are braced or otherwise supported against joint translation. (Bending moments are then taken from within middle 1/3 of beam.) Cw: Warping constant. Simplified warping factor calculated as given in DNVGL-RU-CG-0128, sec 3 [4.4] Table 8 formula Example for I-beam shown below: <table><tr><td></td><td>$I_{sy} = \frac{1}{3} (2b_f t_f^3 + d_{wt} t_w^3) 10^{-4}$</td><td>cm⁴</td></tr><tr><td></td><td>$c_{warp} = \frac{d_{wt}^2 b_f^3 t_f}{24} 10^{-6}$</td><td>cm⁶</td></tr></table>								f_{end}	4.0	2.0	1.0	1.0	0.25	0.25		$I_{sy} = \frac{1}{3} (2b_f t_f^3 + d_{wt} t_w^3) 10^{-4}$	cm ⁴		$c_{warp} = \frac{d_{wt}^2 b_f^3 t_f}{24} 10^{-6}$	cm ⁶
																					
f_{end}	4.0	2.0	1.0	1.0	0.25	0.25															
	$I_{sy} = \frac{1}{3} (2b_f t_f^3 + d_{wt} t_w^3) 10^{-4}$	cm ⁴																			
	$c_{warp} = \frac{d_{wt}^2 b_f^3 t_f}{24} 10^{-6}$	cm ⁶																			
E:	Young's Modulus																				
Poisson:	Poisson's ratio for transverse contraction																				
Yield Stress:	Nominal yield stress																				

Notice that some additional information is given in the 'Pillar' tab in the Properties pane:

Pillar Buckling	
Id	5
Name	
Total length	19750 mm
Elastic length	19750 mm
Buckling length	19750 mm
Braced joint transl.	Yes
Profile	3
Name	VFR
Ax	22500 mm ²
Bx	1,375e+006 mm ⁴
Iy	7,9938e+008 mm ⁴
Iz	1,2631e+009 mm ⁴
Warping fact. Cw	3384745 cm ⁶
Material	3 VL-NS Mild Steel
Yield stress	235 N/mm ²
E	210000 N/mm ²
Poisson	0,3
Strong Axis	
End constr. factor Y	1
Buckling curve Id, Y	curve a
Weak Axis	
End constr. factor Z	1
Buckling curve Id, Z	curve b
Load and stresses	
Acceptance criteria	AC-II
Max allowable UF	0,75

Model / Response / **Pillar**

Table 10 Buckling curves for different cross section

Shape of section	Buckling about axis	Column curve
 Rolled tubes Welded tubes (hot finished)	y-y z-z y-y z-z	a
 y-y: b, t; z-z: h, d Welded box sections Heavy welds (full penetrations) and b/t < 30	y-y z-z y-y z-z	b c
 I and H rolled sections h/b > 1.2 h/b < 1.2	y-y z-z y-y z-z	a b c
 I and H welded sections Flame cut flanges Rolled flanges	y-y z-z y-y z-z	b c c
 I and H sections with welded flange cover plates $t = t_{max}$	y-y z-z y-y z-z	a a
 Box sections, stress relieved by heat treatment	y-y z-z y-y z-z	a a
 I and H sections, stress relieved by heat treatment	y-y z-z y-y z-z	a a
 T and L sections y-y: d z-z: t	y-y z-z y-y z-z	c c
 Channels	y-y z-z y-y z-z	c c

Class guideline — DNV-CG-0128, Edition September 2021
Buckling

Page 51

Section 3



ABOUT DNV

DNV is an independent assurance and risk management provider, operating in more than 100 countries, with the purpose of safeguarding life, property, and the environment. As a trusted voice for many of the world's most successful organizations, we help seize opportunities and tackle the risks arising from global transformations. We use our broad experience and deep expertise to advance safety and sustainable performance, set industry standards, and inspire and invent solutions.