DNV·GL

SESAM USER MANUAL

GeniE

Vol 5 – Code checking of panels

CSR for Bulk Carriers

CSR for Oil Tankers

CSR for Bulk Carriers and Oil Tankers

DNV GL rules

Sesam User Manual

GeniE

Date: 1 Mar 2016

Valid from GeniE version V7.3

Prepared by DNV GL - Software

E-mail support: software.support@dnvgl.com

E-mail sales: software@dnvgl.com

1 CODE CHECKING OF PANELS

This is the user manual for the part of GeniE dedicated to code checking of panels on ship structures according to:

1. **CSR Bulk -** IACS Common Structural Rules for Bulk Carriers.
2. **CSR Tank -** IACS Common Structural Rules for Tankers (using PULS)
3. **CSR BC & OT** (previously also named **CSR Harmonised** in this documentation) IACS Common Structural Rules for Bulk Carriers and Oil Tankers, 1 January 2015 issue, in force July 2015.
4. **DNV GL rules** – October 2015.

This user manual assumes that the user has knowledge in the use of GeniE as covered by the GeniE User Manual Volume I and II – the main user manual and the one describing how to run analysis.

This manual describes how to create a capacity model (or code checking model), how to perform the code checking as well as how to report the results.

Chapter 2 gives an overview of panel checking.
Chapter 3 contains a detailed description on the code checking using CSR Bulk.
Chapter 4 contains a detailed description on the code checking using CSR Tank.
Chapter 5 contains a detailed description on the code checking using CSR BC & OT or DNV GL rules.

1.1 How to read this manual

Read Chapter 2 "*Panel Code checking features of GeniE"* to get an overview on code checking features of GeniE (what you can do).

Read Chapter 3 *"How to do panel code checking – CSR Bulk"* to learn how to set up a capacity model, perform the code checks and how to report the results using CSR Bulk.

Read Chapter 4 *"How to do panel code checking – CSR Tank"* to learn how to set up a capacity model, perform the code checks and how to report the results using CSR Tank.

Read Chapter 5 "*How to do panel code checking – CSR BC&OT or DNV GL rules*" to learn how to set up a capacity model, perform the code checks and how to report the results using CSR BC&OT or DNV GL rules".

A command from the menu list (also referred to as the pulldown menu) is written like this: ***Insert|Plate|Dialog***. The name of a tool button is written like this: **Basic plate**. A function buttons is referred to like this: **F1**.

GeniE comes with a context sensitive menu. You invoke this menu by pushing your right mouse button when the mouse is located above a selected object. In this manual this operation is termed **RMB**. The commands on the context sensitive menu are written like this: *Join Panels*.

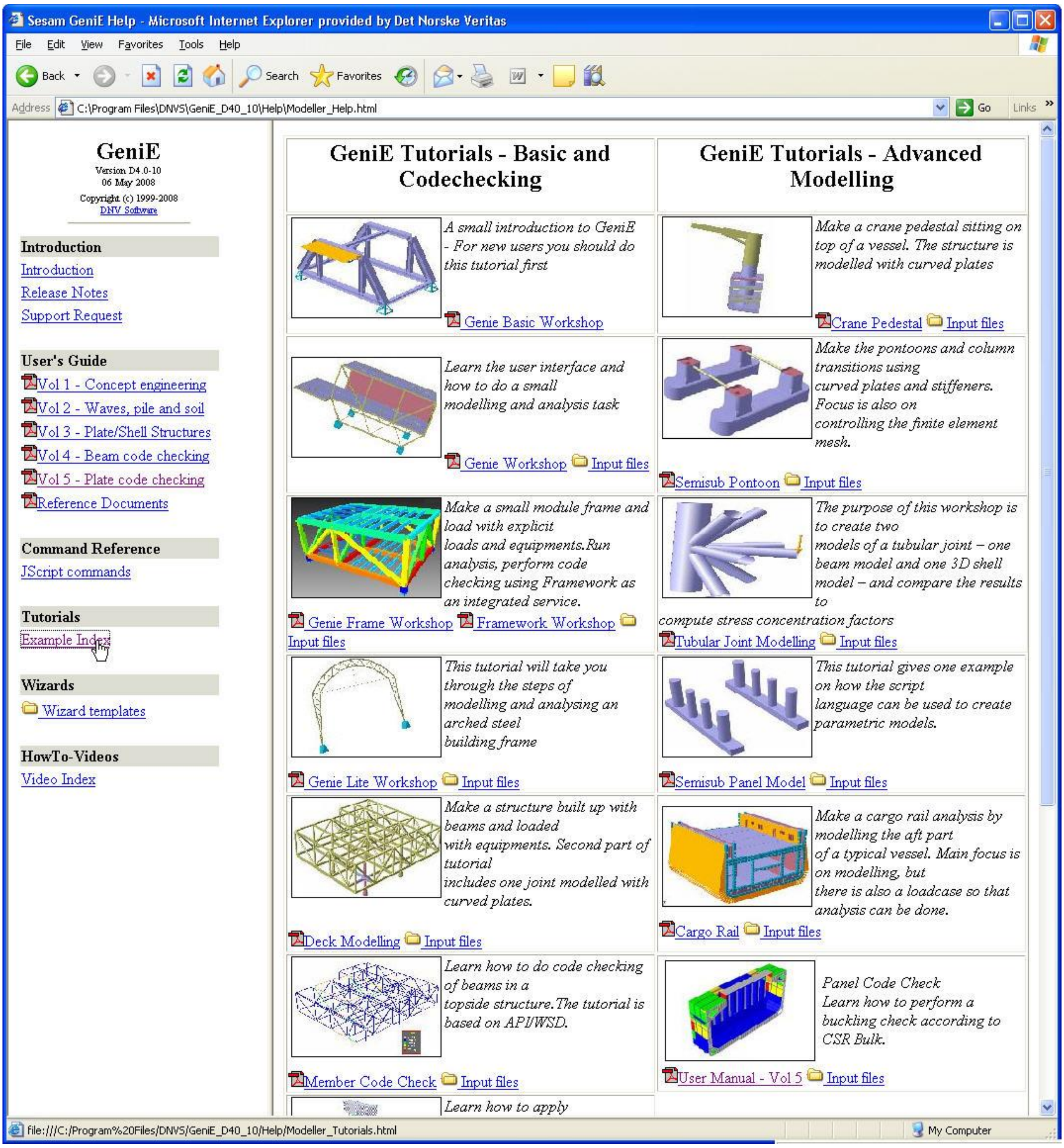
Viewing this manual assumes the usage of Adobe Acrobat Reader version 8.0 or higher. You may use older versions, but then you don't have access to important features like e.g. free text search and bookmarks (table of content + hyperlinks).

It is particularly noted that this User Manual documents all capabilities of GeniE. If you do not have access to the program extension "*Code checking of plates (CCPL)*" there are several items in this manual you do not have access to in your program. These features are blanked out in your program version.

1.2 Learning from tutorials for code checking

GeniE comes with an on-line help system (***Help|Help Topics*** or **F1**). Its purpose is to provide easy access to release notes, limitations, tutorials, wizards and this user manual. In addition it contains a detailed documentation of all available commands in the journaling system (based on J-script). There are also videos showing how to do certain operations, these are best viewed using resolution 1280x1024.

The easiest way to find the tutorials is from GeniE's help page. For panel code checking the most relevant tutorial is called *Panel Code Check.* This is a CSR Bulk tutorial.

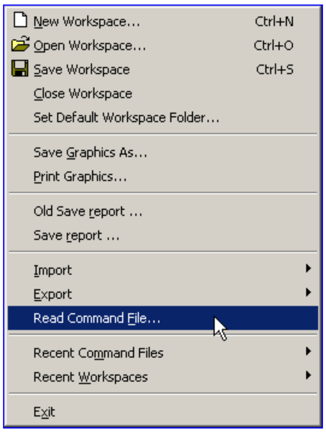


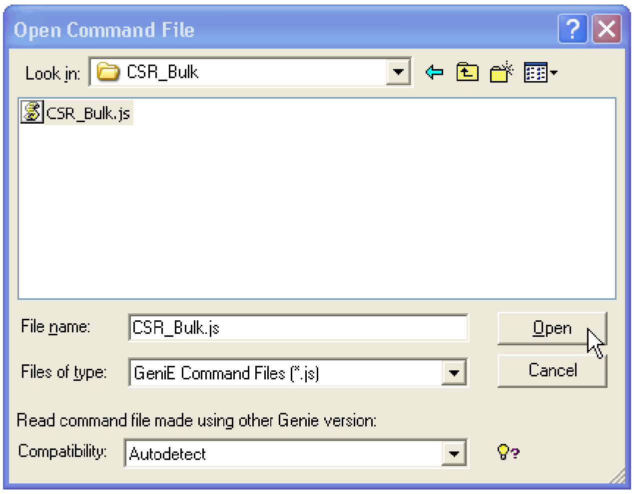
The most efficient way to work with the tutorials is to make a print-out of the tutorials, start GeniE, create a new workspace (command ***File|New workspace***) and follow the steps in the tutorial. Each tutorial comes with a pre-defined journal file (command file) – you find these from the help page as shown above under "Input Files". If you want to use these files the steps are as follows:

1. Create a new workspace ***File|New Workspace|<name>***
   (Keep the default settings for tolerant modelling and database units)

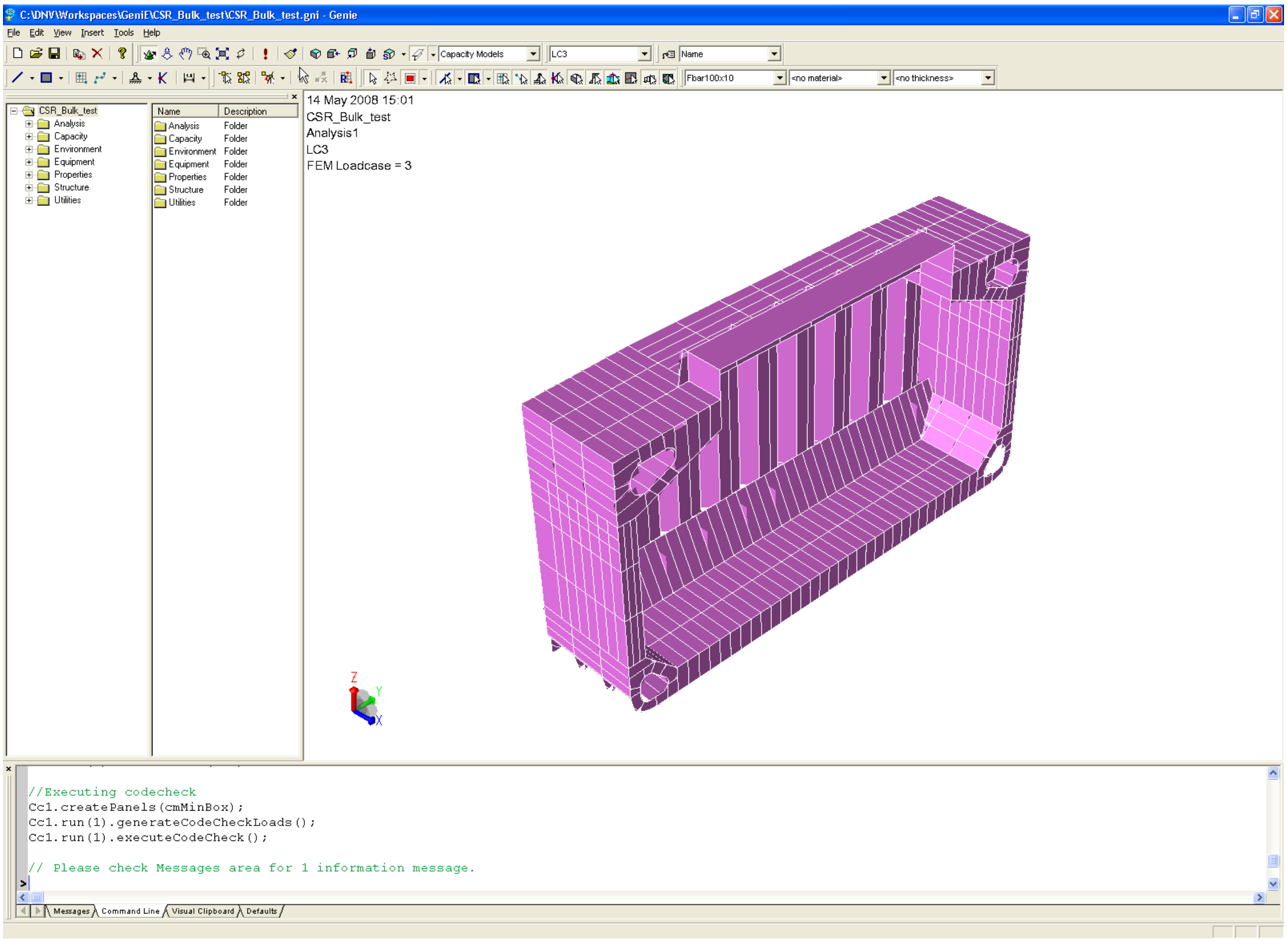
2. Read in the journal file from ***File|Read Command File***|<browse until you find the desired input file>
3. Save your model by ***File|Save***
4. You have now created the same model as in the tutorial you selected.

In the example below the <name> has been set to *CSR_Bulk_test* and the imported file is for the tutorial "CSR_Bulk".





The sequence above creates the following view in GeniE (the colour background has been set to white and the view is set to "Capacity Models":



You may also read in a journal file by using drag-and-drop. Simply select a journal file from your browser and drop it into the command line window.

1.3 Acronyms frequently used in the User Manual

Acronym	Explanation
RMB	Right Mouse Button
LMB	Left Mouse Button
GUI	Graphical User Interface
DOF	Degree Of Freedom
CLI	Command Line Interface
FEM file	SESAM Input Interface File
IACS	International Association of Class Societies
CSR Bulk	Common Structural Rules for Bulk Carriers
CSR Tank	Common Structural Rules for Tankers
CSR BC & OT	Common Structural Rules for Bulk Carriers and Oil Tankers (previously named CSR Harmonised)
PULS	Panel Ultimate Limit State
DNV GL rules	New common ruleset for DNV GL

2 PANEL CODE CHECKING FEATURES OF GENIE

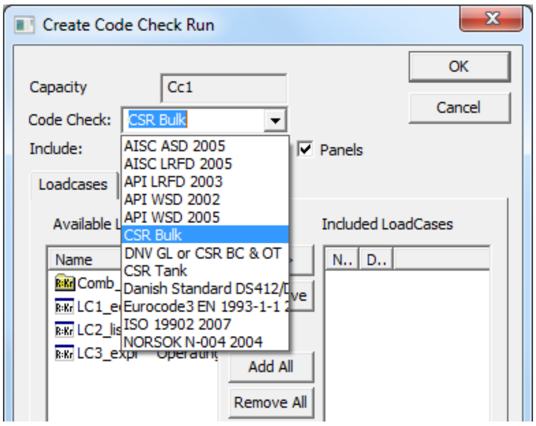
Code checking of panels based on the results from a linear structural analysis can be done in GeniE. The features include checks against allowable stress levels and buckling.

You should be familiar with the relevant rules and procedure of the type of code checking you want to do as this user manual is not intended to cover such. This manual describes how to use Genie to do a panel code check.

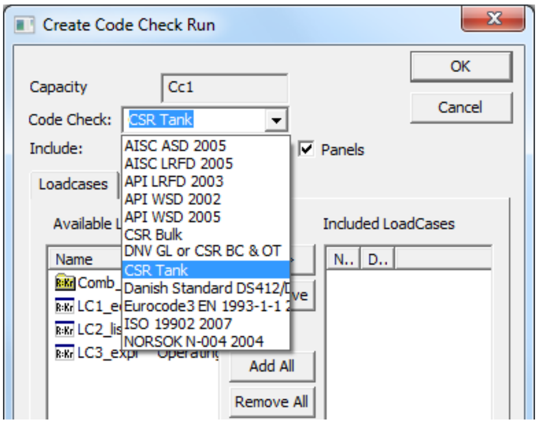
2.1 Panel code checking standards

GeniE supports the following panel code checking standards:

- *CSR Bulk*
  Common Structural Rules for Bulk Carriers, IACS, January 2006

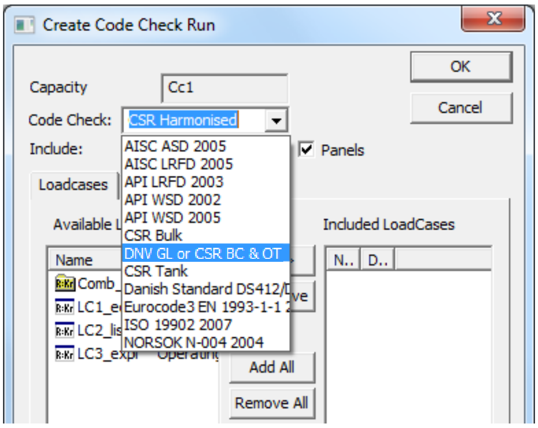


- *CSR Tank*
  Common Structural Rules for Double Hull Oil Tankers with Length 150 Metres and Above, IACS, July 2008.



- *CSR BC & OT*
  Common Structural Rules for Bulk Carriers and Oil Tankers – IACS, January 2015.

- *DNV GL rules*
  DNV GL rules – DNV GL, October 2015.



- Available code checks

The table below shows the type of check that may be performed for each code of practice and the section type that may be processed.

Code of practice	**Check**
CSR Bulk	Panel Yield
	Panel Buckling
CSR Tank	Panel Buckling (PULS)
CSR BC&OT	Panel Buckling (CSR)
	Coarse Mesh Yield Check
	Yield Screening.
DNV GL rules	Panel Buckling
	Coarse Mesh Yield Check

2.2 Code checking

A typical procedure adopted for a panel code check analysis may be as follows:

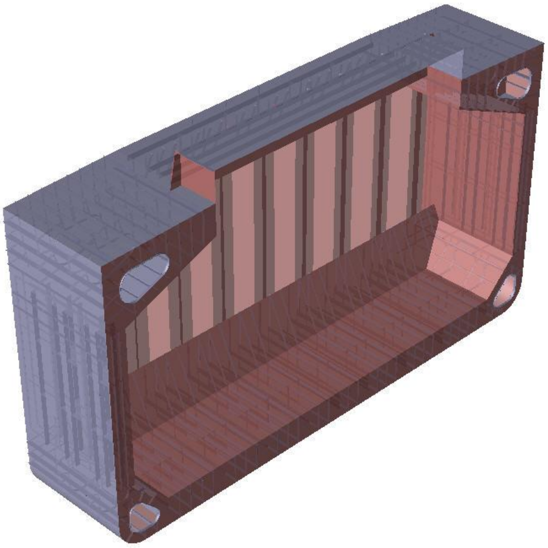
- Modelling of material, thickness, structure, loads and boundary conditions.
- For CSR Bulk and CSR Tank analysis compartments are generated in Genie. Compartment loads, external sea pressure and corrosion additions are provided from Nauticus Hull.
- For CSR Tank stiffener properties must be added (i.e. effective width) must be given in order to proceed for the structural analysis.
- Run the finite element analysis.
- Define relevant load combinations if they were not part of the above analysis.
- Create a capacity manager(s). You decide which analysis you want to base the code checking on. You may have several capacity managers – each capacity manager may have one set of panels. These may be selected from the whole structure or from named sets
- Define the panels.
- Create a code check run. The purpose of this task is to decide which code of practice to use, which loadcases to include and to specify other global factors.
- Compute the code checking forces. These are computed at pre-defined positions.
- Perform the code check and investigate the results graphically or from the browser.
- If necessary modify code checking parameters and re-run. Note that changes made to structure model requires that you re-run the FEM analysis and re-generate the panels. Local changes on some of the capacity panels are allowed, but these will be lost when re-creating the panels.
- Make a report using the report generator and include pictures to it. The granularity of the report depends on how much details you want to add to it. There are several filters you can use to decide the content.

The above procedural steps from creating the capacity manager are illustrated in the following.

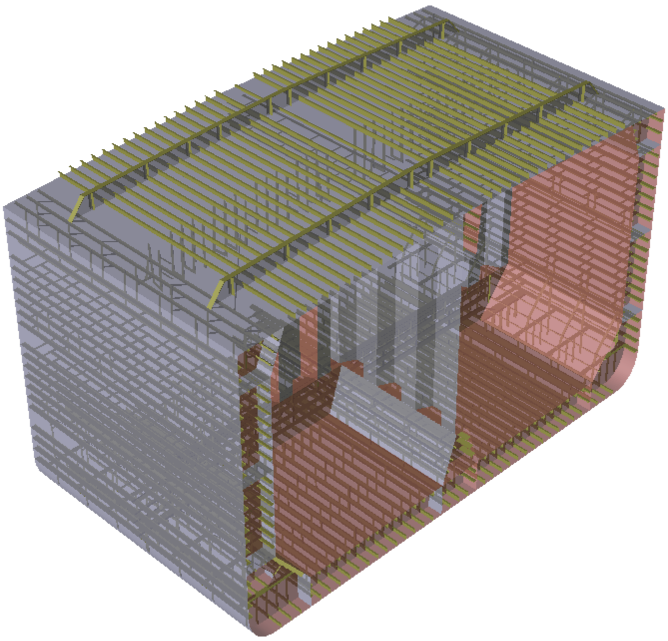
2.2.1 Reference models

Two reference models have been created, one for CSR Bulk and one for CSR Tank.

The CSR Bulk model is a small slice from the three cargo hold model from the CSR Bulk tutorial.



The CSR Tank model is a small slice of a product tanker. The model includes a corrugated transverse bulkhead and a few frames.



The purpose of the following tutorial is to create a capacity model to do panel check. The reference models have one analysis activity and a few simple loadcases.

2.2.2 Loadcases

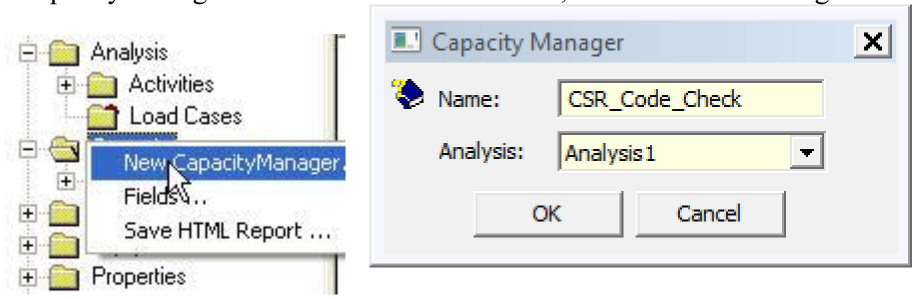
The reference models have a number of basic loadcases (manually applied) and load combinations.

2.2.3 Create a capacity manager

The purpose of a capacity manager is to decide which analysis results to use in the code checking. You may have several capacity managers if you want to use

- various analysis results
- different selection of capacity panels

A capacity manager is created from the browser, and in this case it is given the name "CSR_Code_Check".

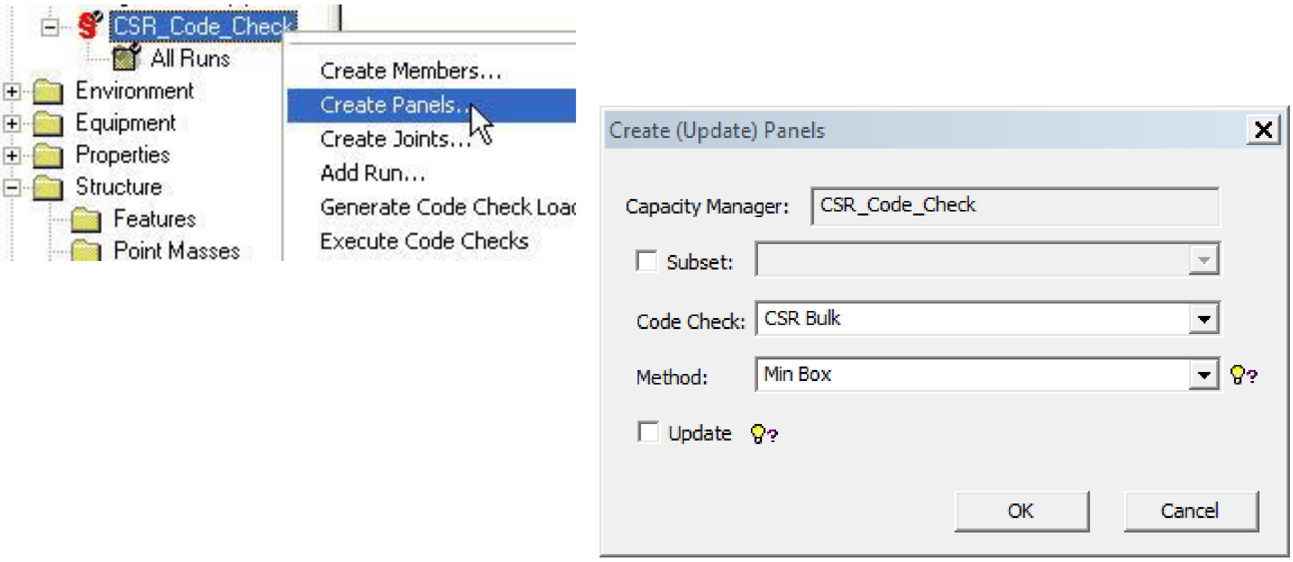


2.2.4 Create capacity panels

We now need to define the panels. In a code check the panels are normally not the same as the concept model, hence it is necessary to define the capacity panels by splitting the concept plates.

The user must select the appropriate rules, and only relevant options according to the selected rules will show up.

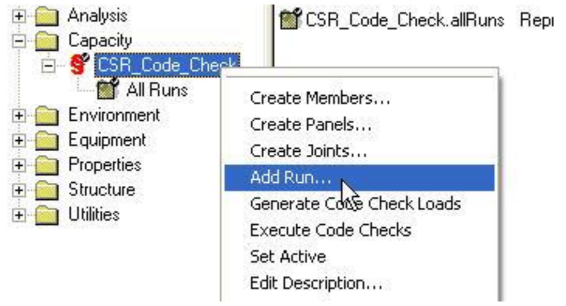
The capacity panels can be visualised by using the view setting Capacity Model



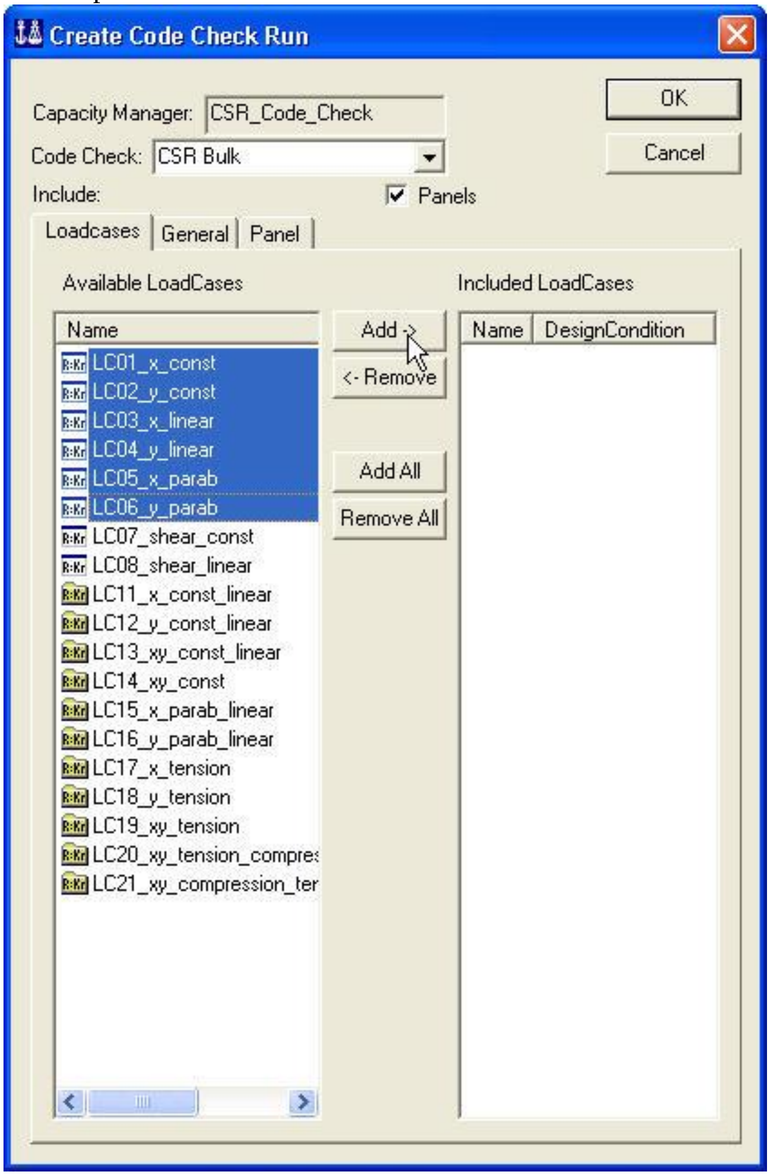
2.2.5 Create a CSR Bulk code check run

When you define a code check run you decide which code of practice to use. We here select the CSR Bulk code check.

For information on how to make a CSR Tank run, see chapter 4 - *How to do panel code checking – CSR Tank.*

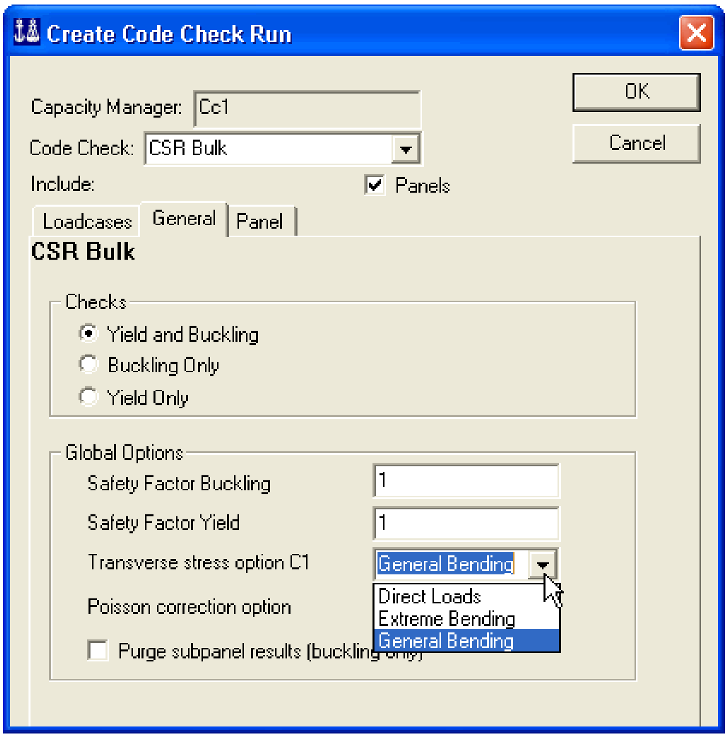


You need to select which LoadCases you want to include in your run. This is done by selecting the LoadCases and clicking Add/Add All.



2.2.6 Modifying the General settings in the CSR Bulk code check run

There are several general settings you can alter for the run. Normally you would not alter the settings, but sometimes it may be useful. Splitting yield and buckling into separate runs makes it simpler to evaluate results.



Checks:

Yield and Buckling - For checking both for yield and buckling. This is set as default.

Buckling Only - For checking for buckling only.

Yield Only - For checking for yield only

Often you would check for both buckling and yield in the same run, but split if you want to assess yield and buckling results separately.

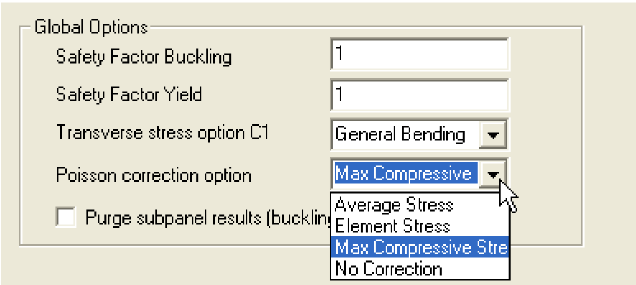
Global Options:

Safety Factor Buckling and Safety Factor Yield - You can set the safety factors larger than 1 if you want the construction to have a lower allowed usage rate. A higher safety factor gives a "safer" structure.

Transverse stress option C1

See CSR Bulk rules, chapter 6.3, table 2 for an explanation. Three alternatives are available:

- Direct Loads
- Extreme Bending
- General Bending – This is default.



Poisson correction option

Four alternatives are available:

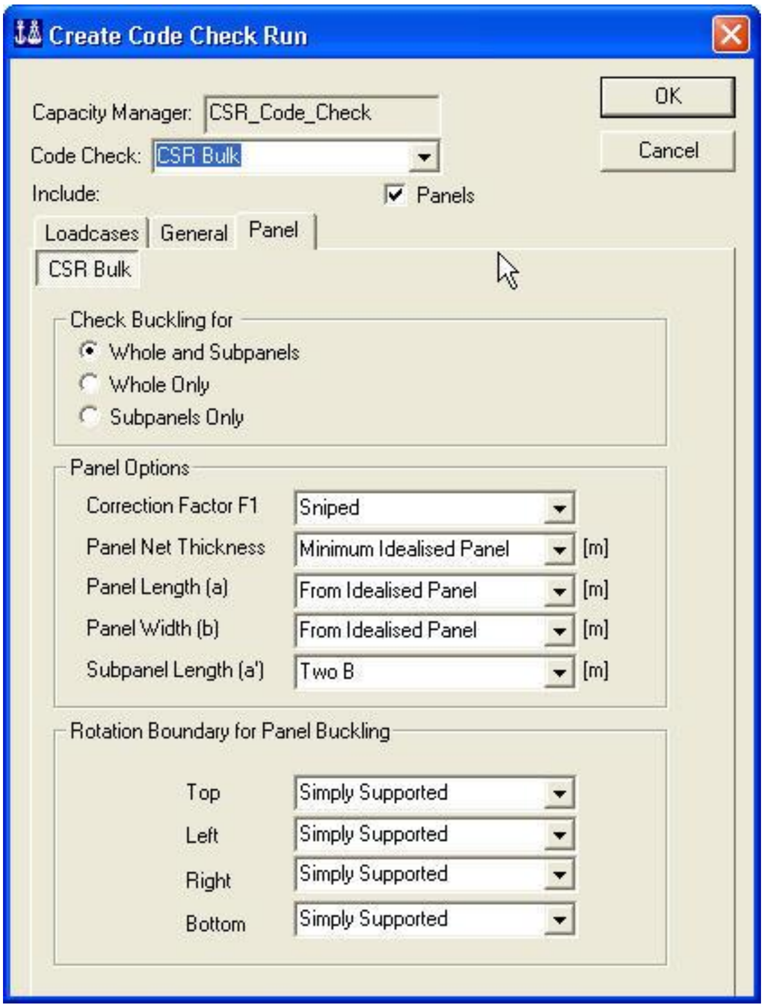
- Average Stress – Uses the average X-stress and average Y-stress as input to the Poisson correction for the whole panel.
- Mac Compressive Stress – Uses the max compressive X-stress and max compressive Y-stress as input to the Poisson correction for the whole panel. - This is default.
- Element Stress – Performs the Poisson correction element by element.
- No Correction – This option is primary available for parameter studies. It will give higher usage factors than when the Poisson correction is applied for bi-axial compression.

Purge subpanel results (buckling only)

With this option checked, Genie will still perform buckling check for all subpanels, but will only store results for the one with the highest usage factor.

2.2.7 Modifying the Panel settings in the CSR Bulk code check run

There are several panel settings you can alter for the run. Normally you would not alter the settings, but sometimes it may be useful.



Check Buckling for:

Whole and Subpanels – For checking buckling both on the panels and on the subpanels. All panels that are long compared to the breadth (aspect ratio greater than 3) are split into subpanels.

Whole only – for checking buckling only on whole panels, not taking subpanels into consideration.

Subpanels only – for checking buckling only on subpanels.

Panel Options:

Correction Factor F1 – Correction factor for boundary condition of stiffeners on the longer side of panels as described in chapter 6.3 in the CSR Bulk rules.

Panel Net Thickness – you can manually insert a net thickness or use the default net thickness.

Panel Length (a) – Length of elementary plate panel

Panel Width (b) – Breadth of elementary plate panel

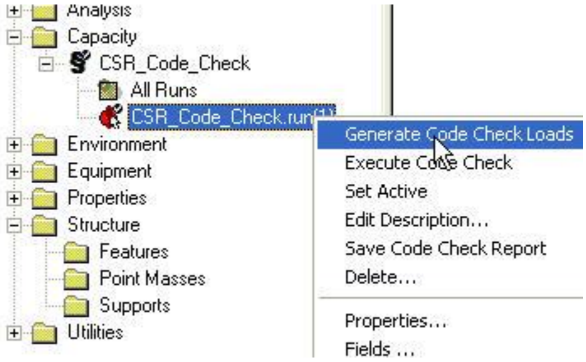
Subpanel Length (a') – Length of subpanel. One panel can consist of several subpanels if its length is long compared to its breadth. Default subpanel length is 2 x b.

Rotation Boundary for Panel Buckling:

Normally, all edges are assumed simply supported, you can however override this by freeing or clamping some of the edges. See CSR Bulk rules, chapter 6.3 table 2 for details.

The CSR Bulk code check run is now available in the browser – the browser will be used when looking at the code checking results later.

2.2.8 Compute code checking forces

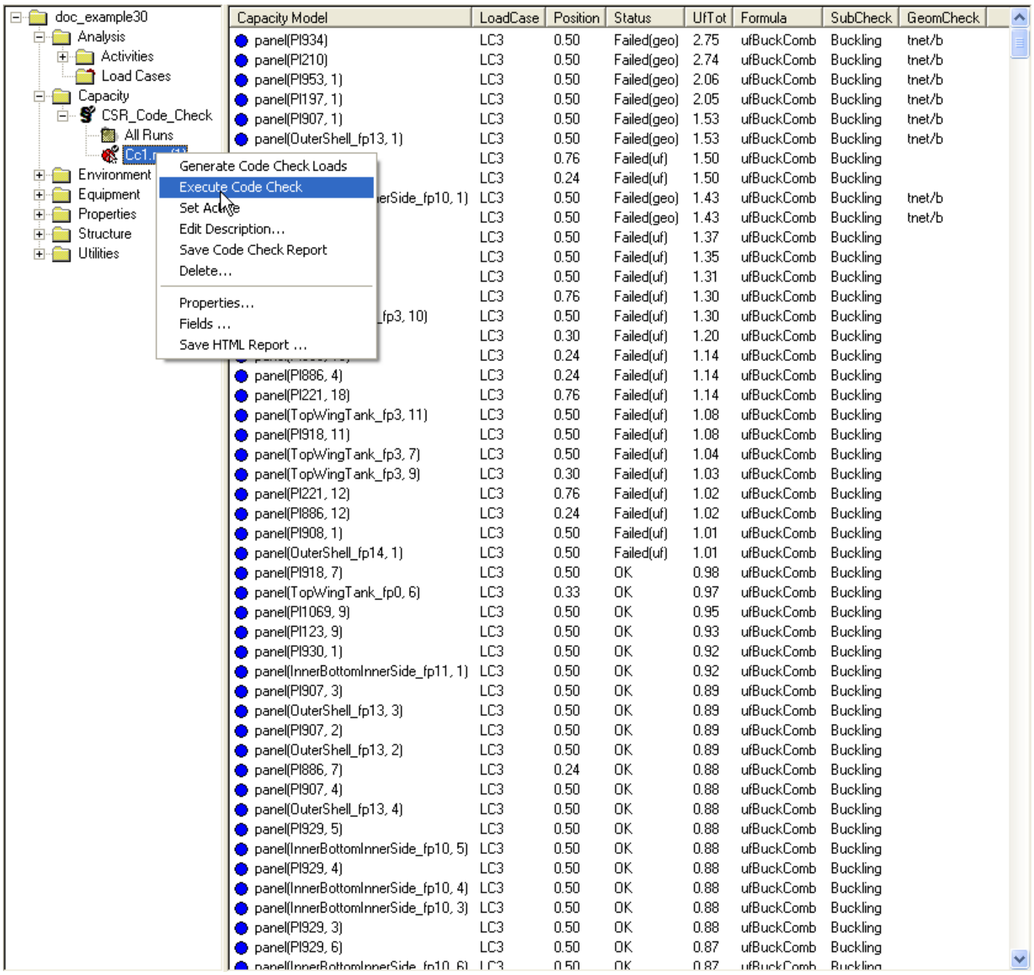


The finite element analysis will compute the element stresses within each finite element. For the code checking it is necessary to compute the forces for the idealised panels. The code checking positions are automatically determined by GeniE. Whenever the concept model has been updated, and the finite element analysis has been reexecuted, the code checking forces must be recomputed.

The code checking forces are computed by selecting an individual run or a capacity manager in the browser. Click RMB and select *Generate Code Check Loads*.

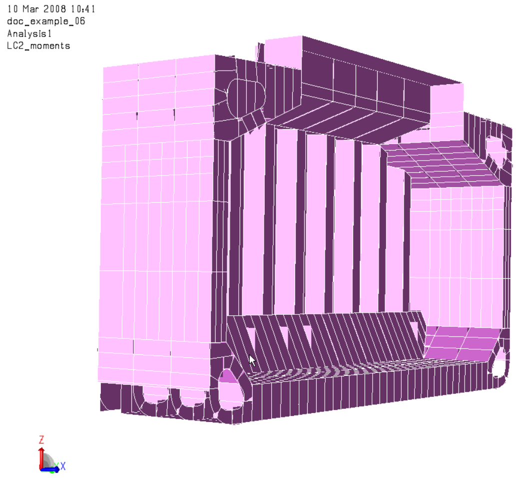
2.2.9 Perform the code check

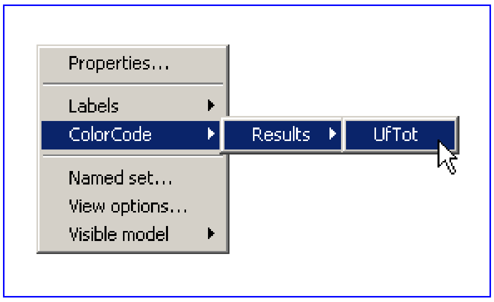
The code checks for panels can now be done from the browser (remember to specify which code check run is active). To the left is shown how to start the code check and the associated results in the browser and a colour coded view.

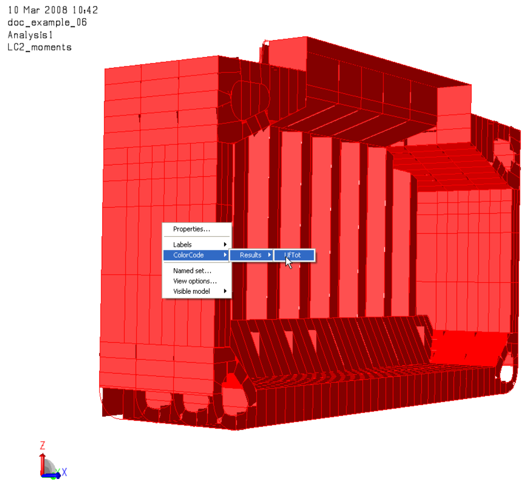


If you have several code check runs, you may compute the forces and execute the code checks for all your runs from the "All Runs" folder.

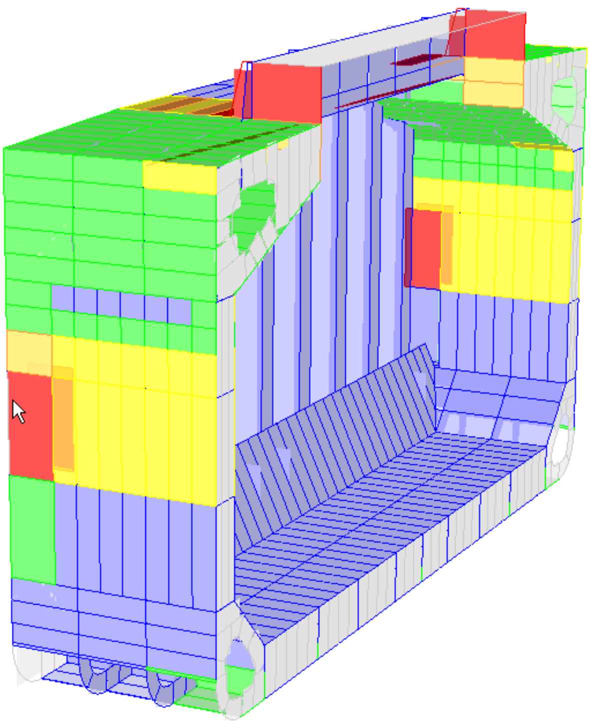
To make a colour coded view of the utilisation factors, simply select all capacity panels in the capacity view, RMB and select *ColorCode|Results|UfTot*.







On the illustration below, the transparency levels are set higher for panels with low usage factor. This makes it easier to see the problem areas.

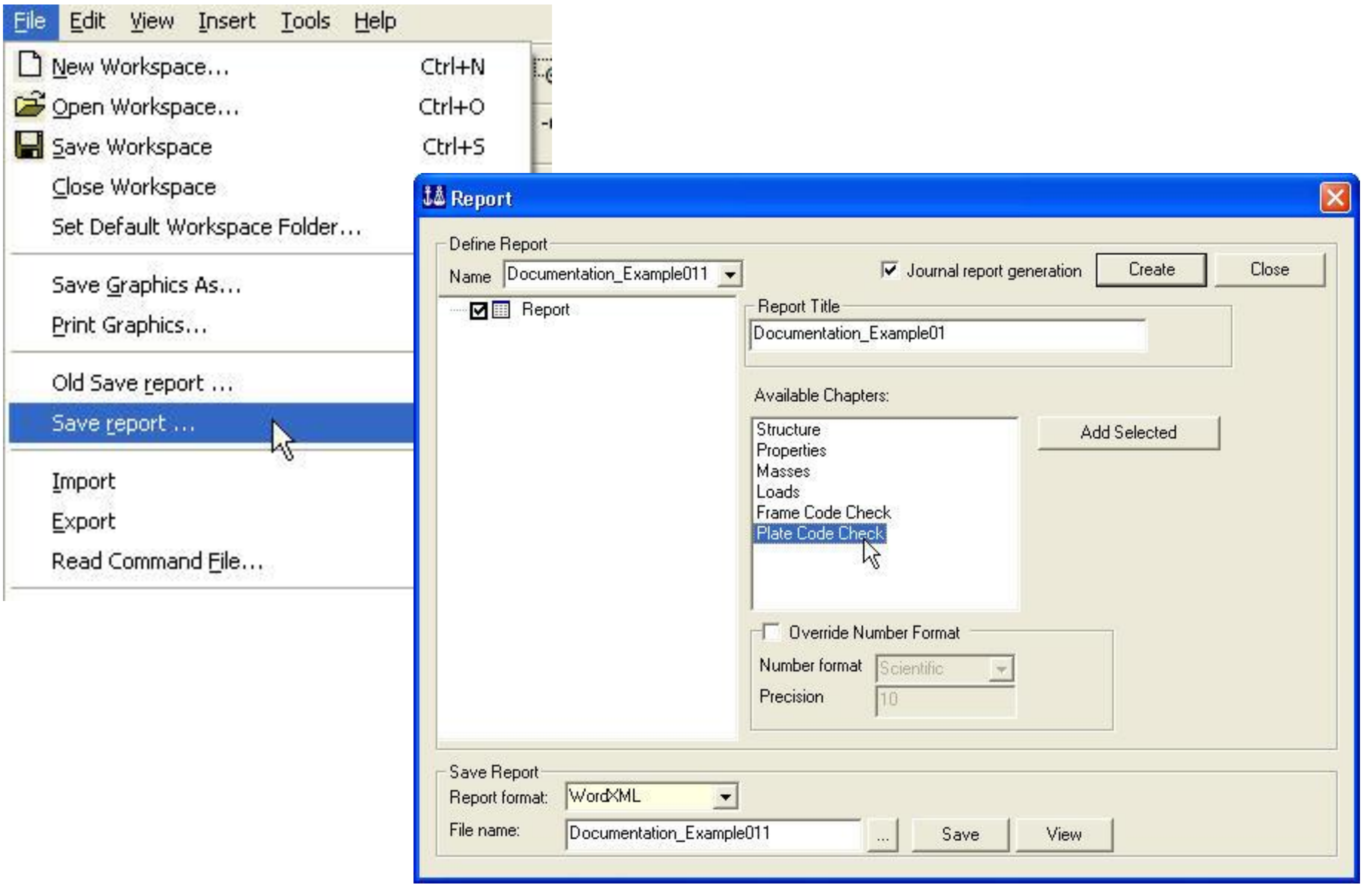


UfTot

- < 0.01
- >= 0.01
- >= 0.5
- >= 0.8
- >= 1
- >= 1.33

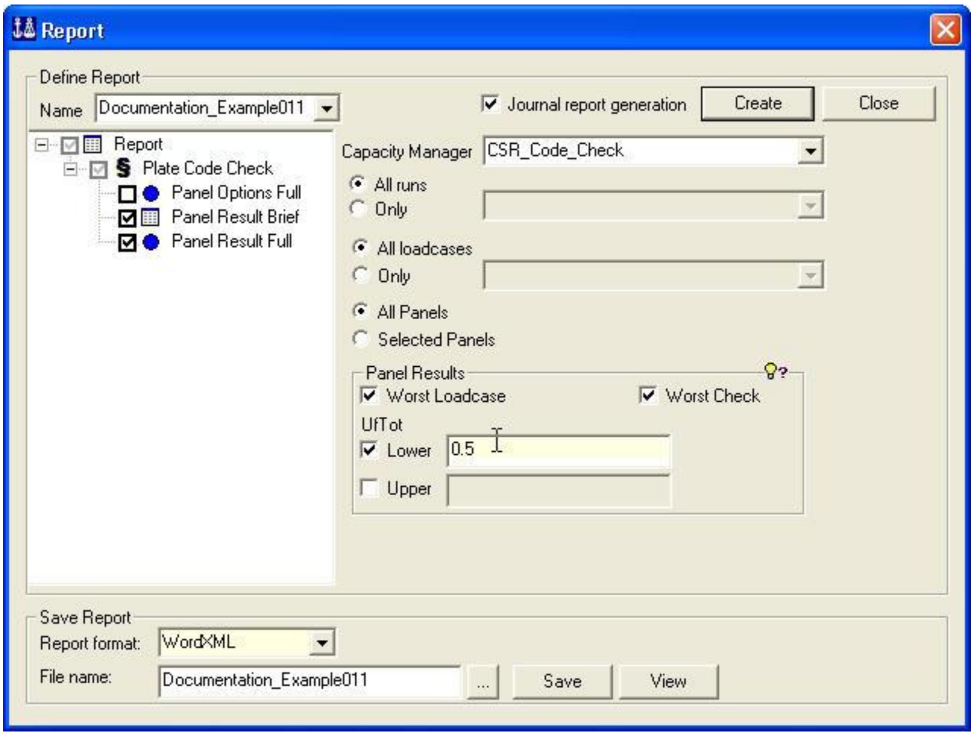
2.2.10 Making a code checking report

The report generator comes with templates to easily generate reports for viewing in text format (typically by using MS Notepad), html format (typically from an internet browser), spreadsheet (MS Excel) or formatted report (MS Word).You may use filters to limit the amount of data that is reported. Furthermore, your report settings may be saved so that you easily can recreate the same report in another code checking run.



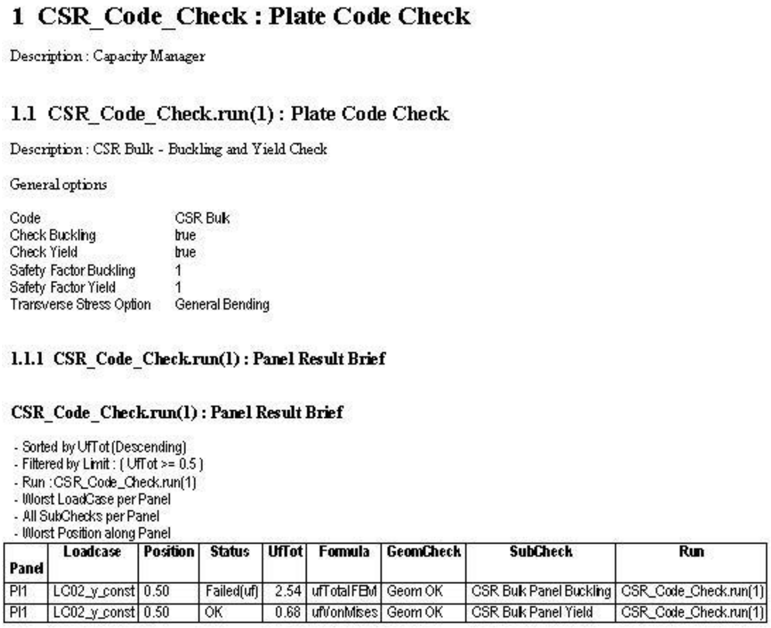
When you have added the Chapter “Plate Code Check” to your report it is possible to use the filters to specify the content you want.

This example uses the default report settings except for the limit of 0.5 for the lower utilisation factor.



SESAM GeniE D4.0-03	Report: Documentation_Example011	Model Id: Documentation_Example011		Sign: atrue
		Description: Documentation_Example01		Date: 05-Mar-2008
		Model file name: C:\DNV\Workspaces\GeniE\Documentation_Example01		Last saved: 05-Mar-2008 15:23:38

A typical page of a report when viewing it in MS Word is shown to the right.



1 CSR_Code_Check : Plate Code Check

Description : Capacity Manager

1.1 CSR_Code_Check.run(1) : Plate Code Check

Description : CSR Bulk - Buckling and Yield Check

General options

Code	CSR Bulk
Check Buckling	true
Check Yield	true
Safety Factor Buckling	1
Safety Factor Yield	1
Transverse Stress Option	General Bending

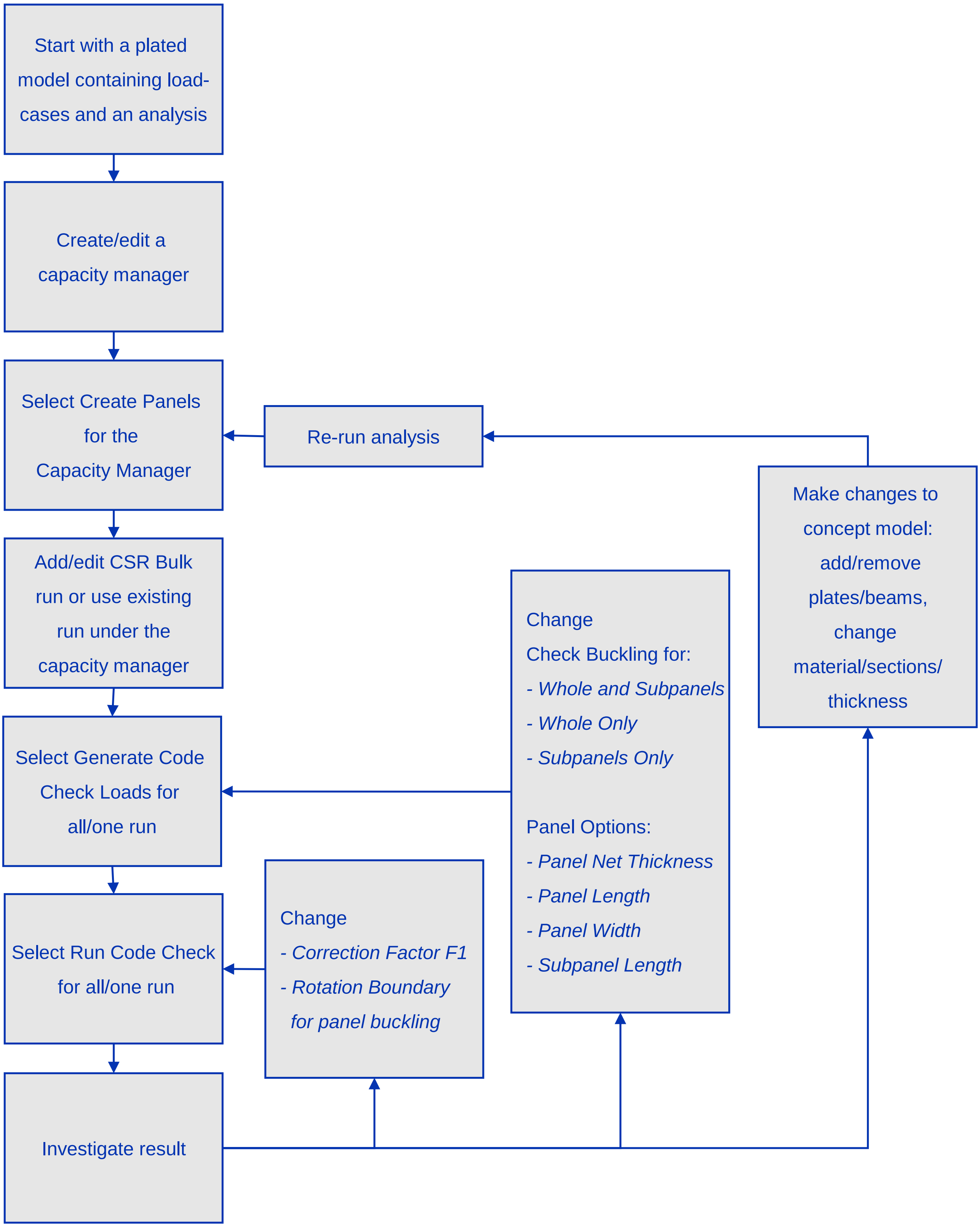
1.1.1 CSR_Code_Check.run(1) : Panel Result Brief

CSR_Code_Check.run(1) : Panel Result Brief

- Sorted by UfTot (Descending)
- Filtered by Limit : (UfTot >= 0.5)
- Run : CSR_Code_Check.run(1)
- Worst LoadCase per Panel
- All SubChecks per Panel
- Worst Position along Panel

Panel	Loadcase	Position	Status	UfTot	Formula	GeomCheck	SubCheck	Run
Pl1	LC02_y_const	0.50	Failed(uf)	2.54	ufTotalFEM	Geom OK	CSR Bulk Panel Buckling	CSR_Code_Check.run(1)
Pl1	LC02_y_const	0.50	OK	0.68	ufVonMises	Geom OK	CSR Bulk Panel Yield	CSR_Code_Check.run(1)

2.3 The work flow of a panel code check



3 HOW TO DO PANEL CODE CHECKING – CSR BULK

This Chapter will guide you through the steps which are necessary to do a CSR Bulk panel code checking. A reference case will be used; the slice of a bulk ship that was introduced earlier in this user documentation.

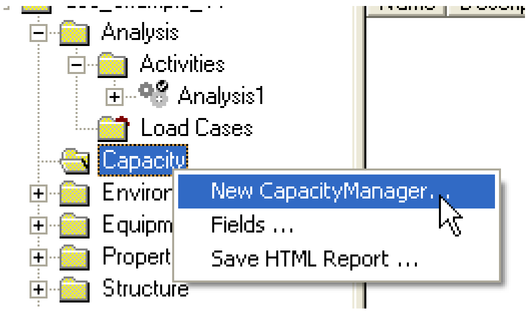
The following procedure may be used when performing code checks; each is described in detail except for the four first steps (please consult the User Manual Vol. I for guidance).

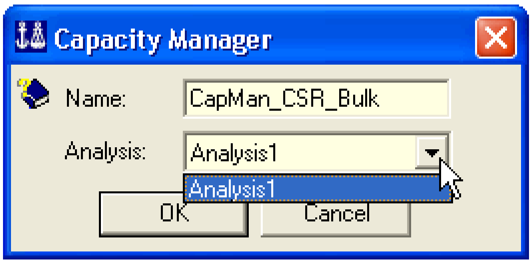
- Make a plate model. Beams may be included. A "real world" model will typically contain both beams and plates.
- Run the finite element analysis
- Define relevant load combinations if they were not part of the above analysis.
- Create a capacity manager
- Define the panels
- Create a code check run
- Assign specific settings to individual panels
- Compute the code checking forces
- Perform the code check and investigate the results graphically or from the browser.
- If necessary modify plate thickness, materials or other code checking parameters and re-run.
- Make a report

3.1 Create a capacity manager

The purpose of a capacity manager is to decide which analysis results to use in the code checking. It is possible to have several analysis activities in GeniE where you can have different loadcases as well as analyse subsets of your model. To be able to do code checking you need to define multiple capacity managers referring to the different analysis in question.

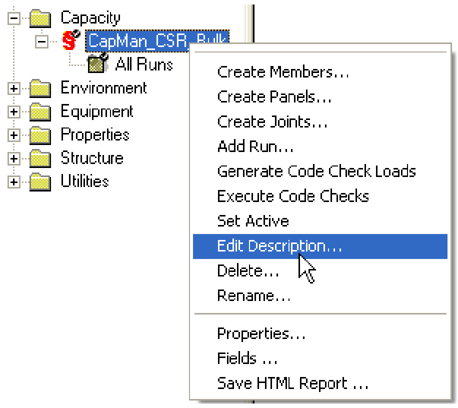
In this case we have one analysis, and one capacity manager is created from the browser.

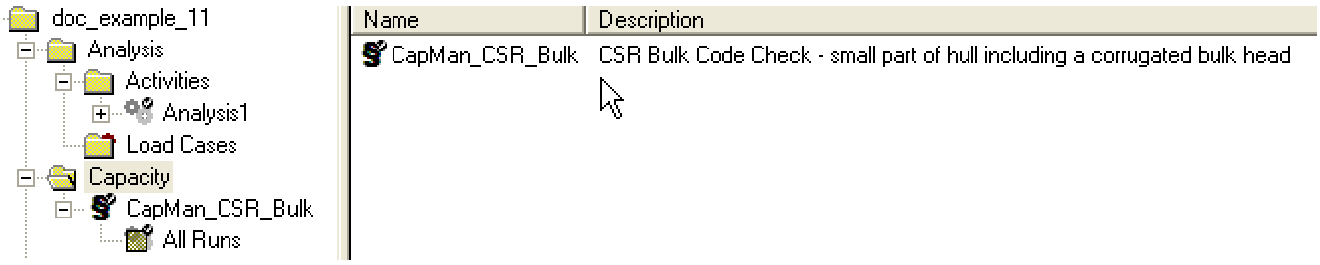




You may add additional description to the capacity manager. The description is also shown in the browser.

When making a report, the descriptions are also documented.

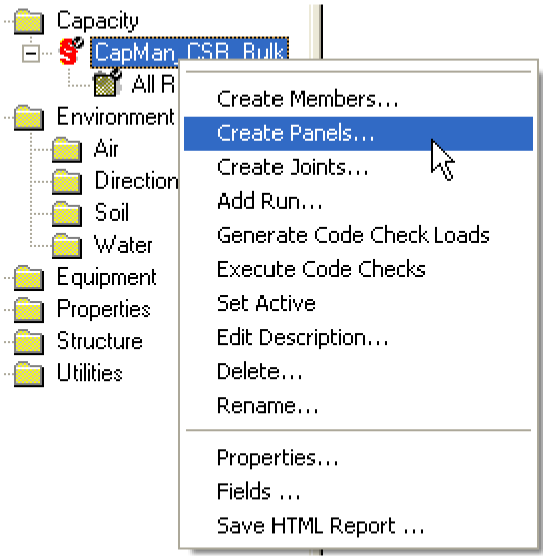


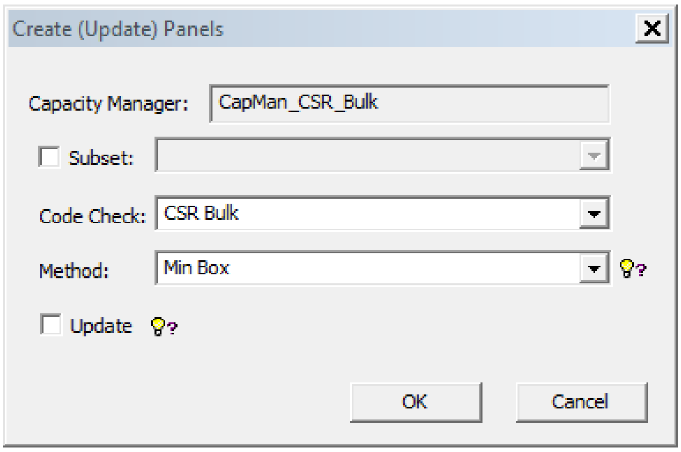


3.2 Define panels

When modelling a concept model it is possible to make continuous plates that span several beams. This means that the concept model is different from a panel capacity model which spans between two beams only. It is therefore necessary to split up the concept model into an elementary panel capacity model.

When the model is split or kept, the default buckling lengths are set since they are the same as the length or the breadth of a capacity panel.





If you are working on a large bulk carrier model it is necessary to divide the model into smaller subsets and create panels for one subset at a time. This can be done by checking the *Subset* check button in the “Create Panels” dialog and choose a subset from the subset list.

When creating panels there are 2 choices available from GUI:

- **MinBox:** finds the smallest idealised rectangular panel possible enclosing the possibly non-rectangular structural region. This method should be used when preparing a CSR Bulk codecheck. Will create planar panels.
- **MaxAreaMoment**: Is an alternative algorithm. Will create planar panels.

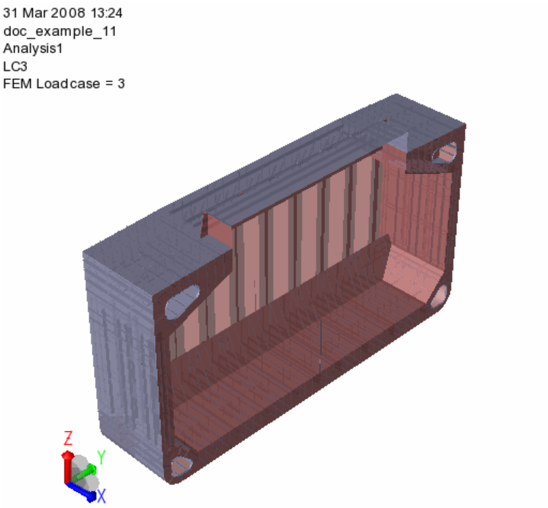
Note that the **CSR_BC_OT_Default** method also will create panels for curved regions. However, this method should not be used for subsequent execution with the CSR Bulk or CSR Tank code-check, since these code-checks in GeniE do not include specific computations for panels created from curved regions. If however, this method is used with CSR Bulk or CSR Tank code-check, the validity of the results must be evaluated by the user – the more panel curvature increases the more non-consitent results.

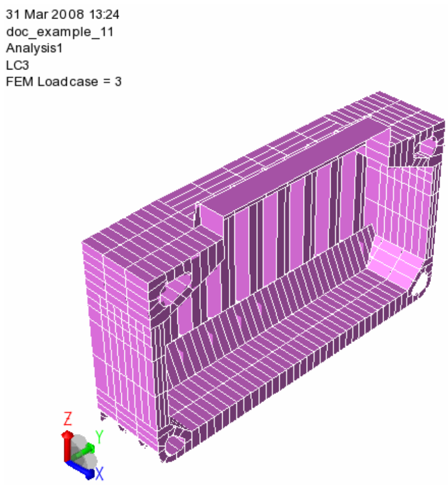
The other panel creation methods will only accept slighty curved regions, within 1 degree of curvature, and only create planar panels.

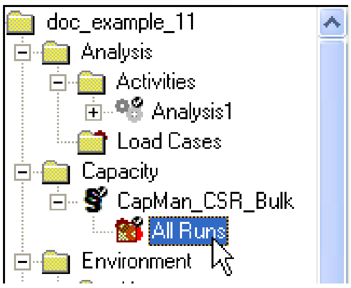
It most cases **MinBox** can be used and it is therefore set as default.

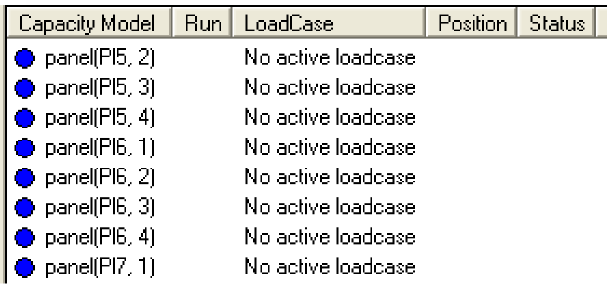
The naming convention of capacity panels refer to the plates. If Plate *Pl20* is split into two capacity panels they are denoted *panel(Pl20,1)* and *panel(Pl20,2)*.

The pictures show the concept model and the capacity model.









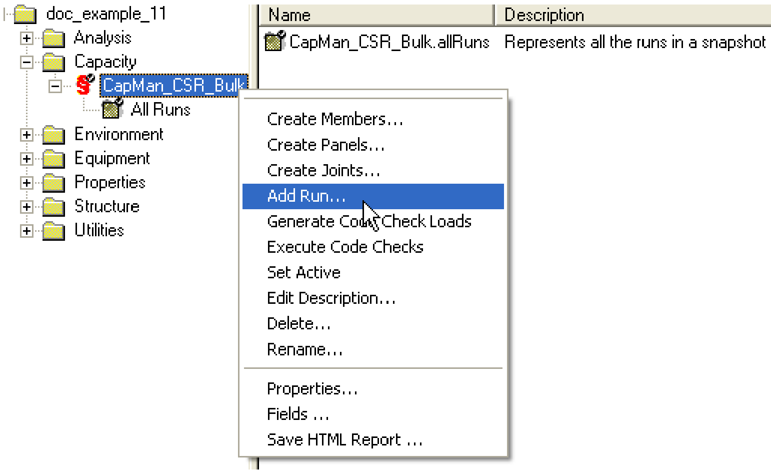
The browser lists each capacity panel. If you have several capacity managers you need to specify which manager is active (select a manager, RMB and choose *Set Active*). In this case “CapMan_CSR_Bulk” is the only capacity manager and it is set to active

3.3 Create a code check run

During the definition of code check runs you decide

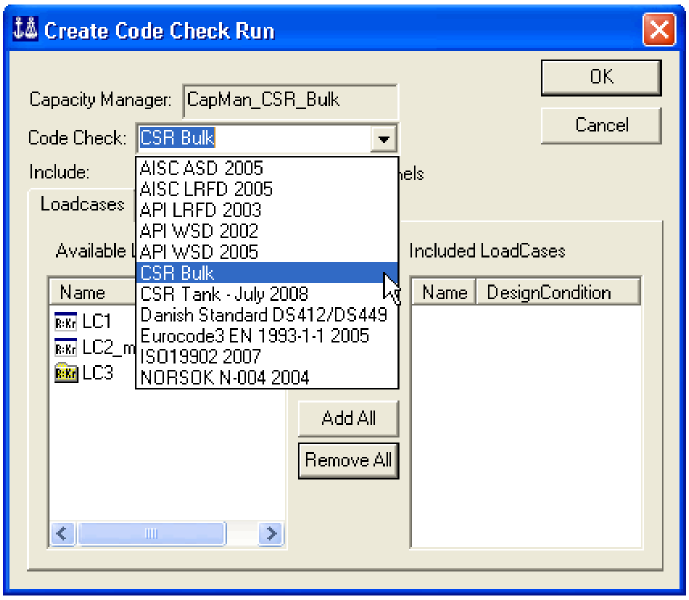
- which code of practice to use
- which loadcases to use
- global code checking parameters (i.e. those who apply to the entire capacity model) – for example buckling lengths or safety factors

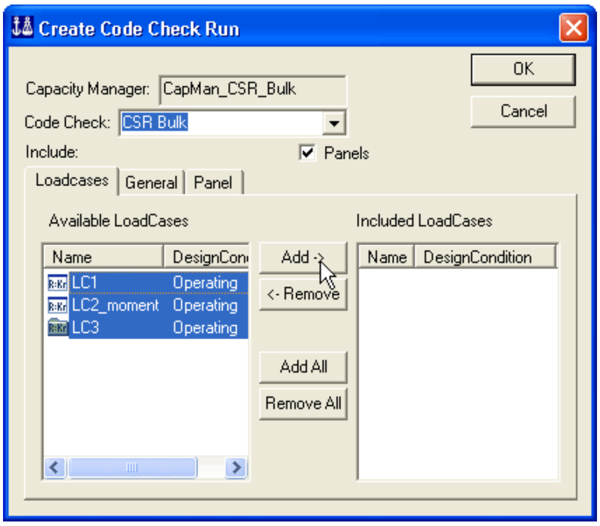
The code check run is defined from the browser.

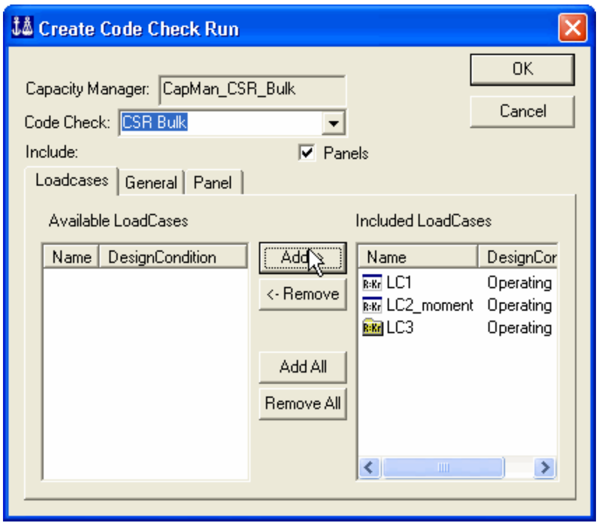


In the example to the right the code check CSR Bulk has been selected.

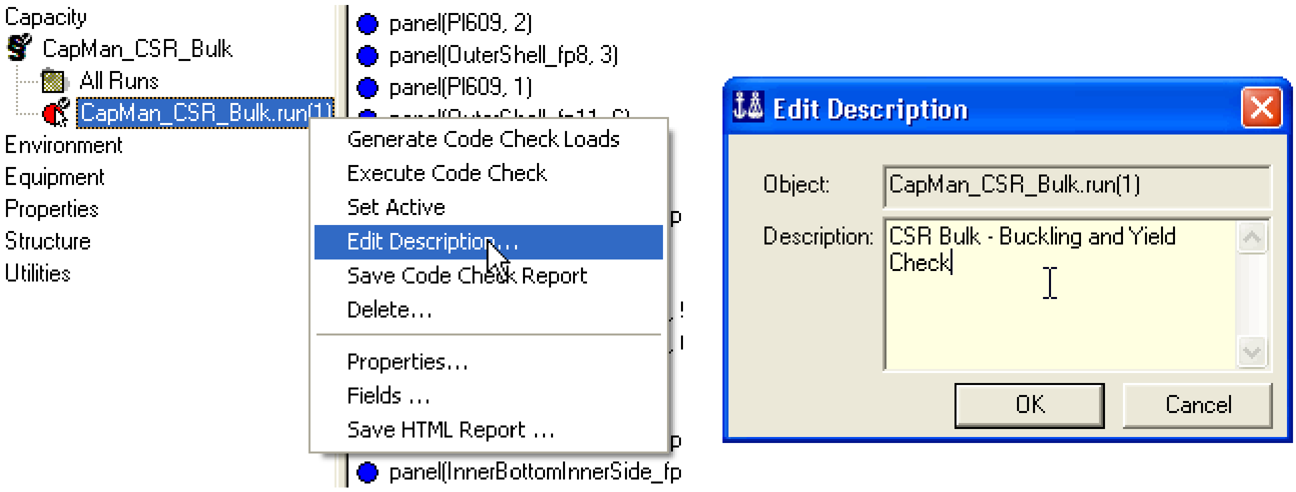
Furthermore, all the loadcases have been added to the code check run.







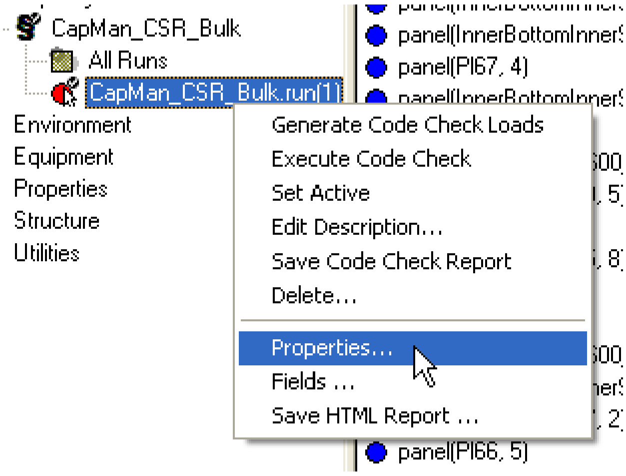
You may modify the code check run from the browser and define additional information to the run.



The same procedure may be used to create another code check run. You then have to specify which one is active from the browser.

3.3.1 Define global general code checking parameters

You specify and modify the general code checking parameters when you define the code check run, or you may modify at a later stage. To modify, select *Properties* as shown on the picture to the right.

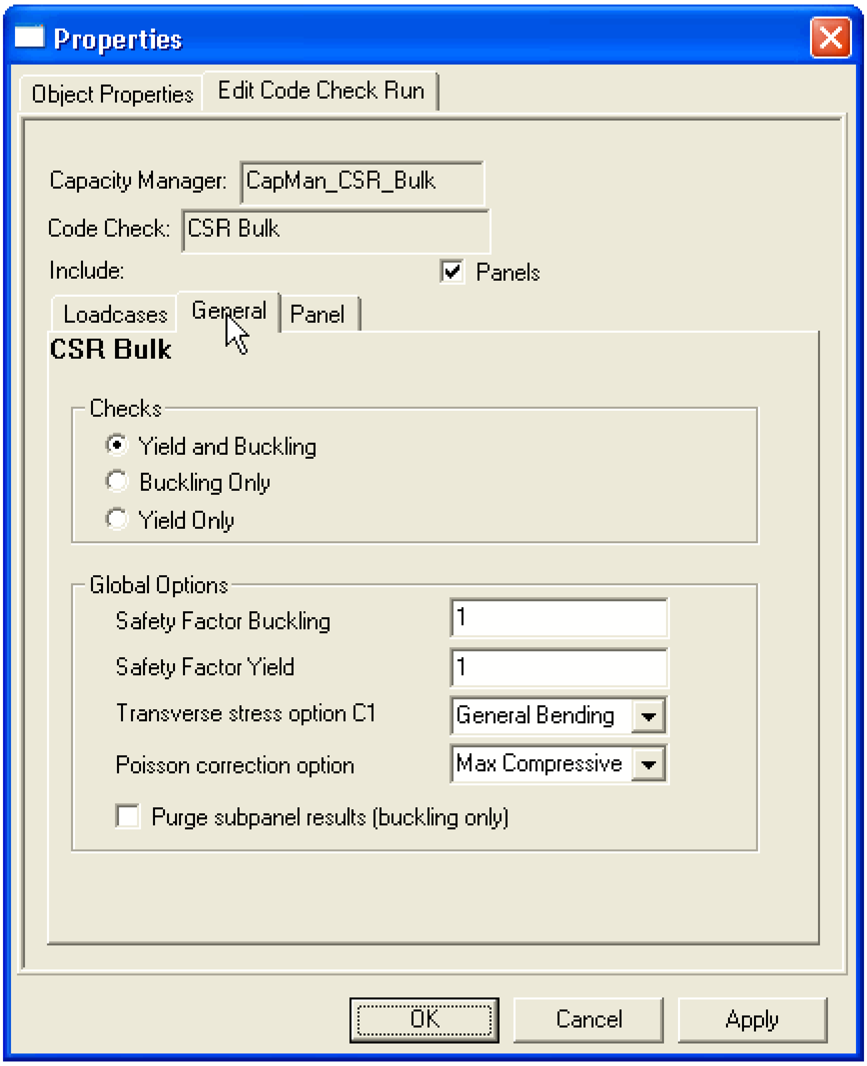


For a further description of parameters shown herein, please consult the relevant codes of practices.

The general code checking parameters are global, i.e. they apply to all capacity panels.

3.3.1.1 General parameters CSR Bulk

The general parameters for the offshore code check CSR Bulk are shown to the right. The parameters are explained in chapter 2.2.6.

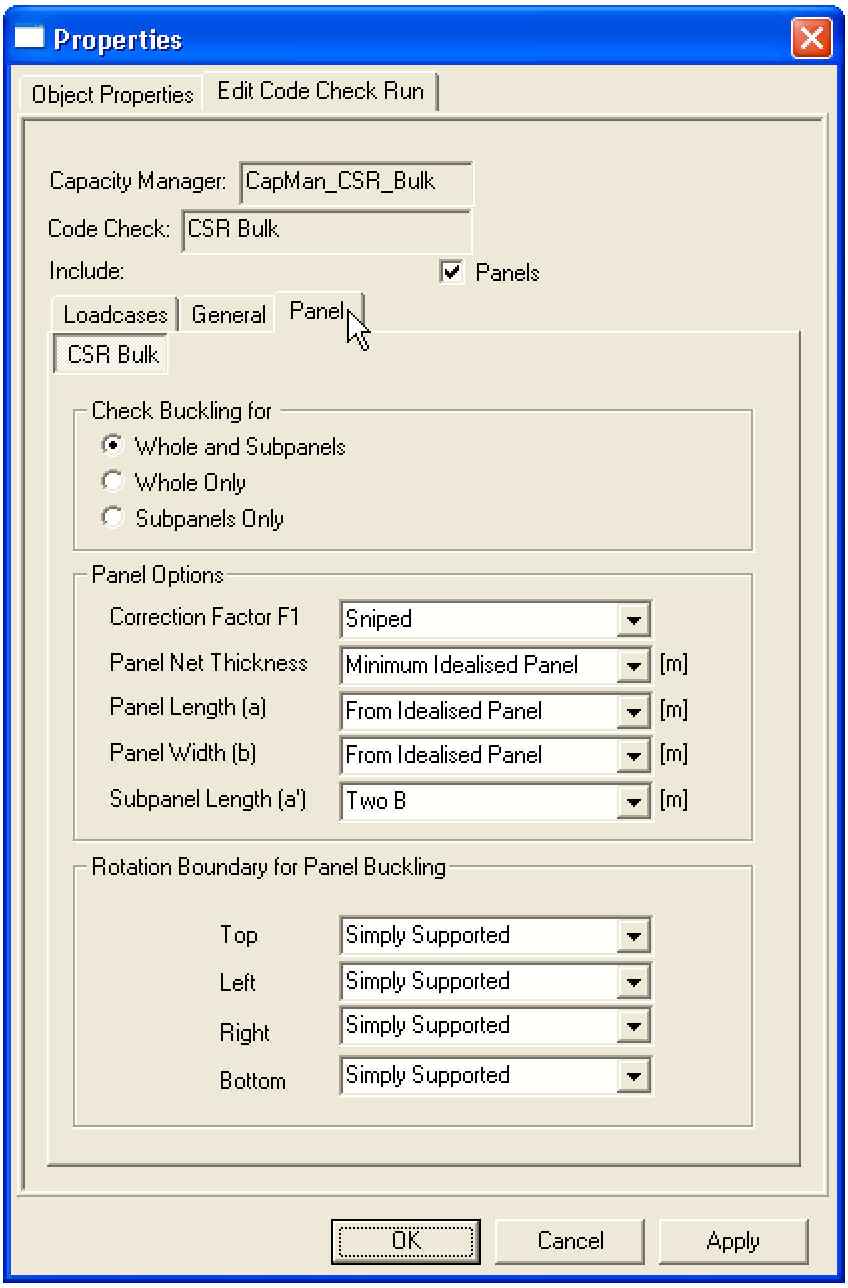


3.3.2 Define global panel parameters

The global panel parameters (those who apply to the all capacity panels) may be changed from default values when you define the code check run or later. The default values are shown in the following for the CSR Bulk code check alternative.

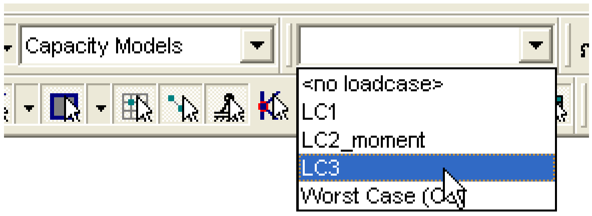
3.3.2.1 Panel parameters CSR Bulk

The default data for panels are shown to the right. The parameters are explained in chapter 2.2.7.

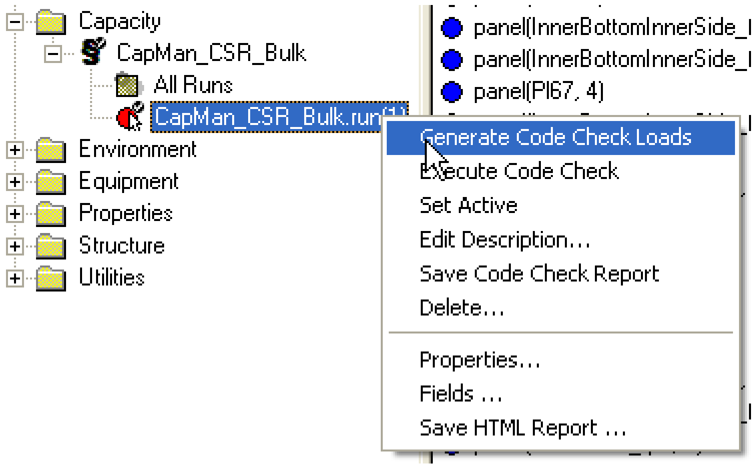


3.4 Perform the code check

To do the code check, we first have to select a loadcase that is included in our run. To do this, select the loadcase from the combo box on top of the screen, like shown to the right.



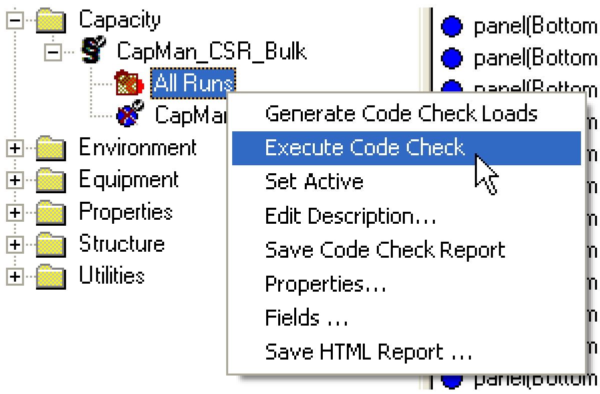
We then have to generate the code check loads. This can be done for one run individually, or for all the runs at the same time. To generate code check loads for all the runs, click **RMB** at *All Runs* and select *Generate Code Check Loads.* To generate code check loads for one individual run, click **RMB** on the run of interest, and select *Generate Code Check Loads.* This is shown in the illustration to the right.



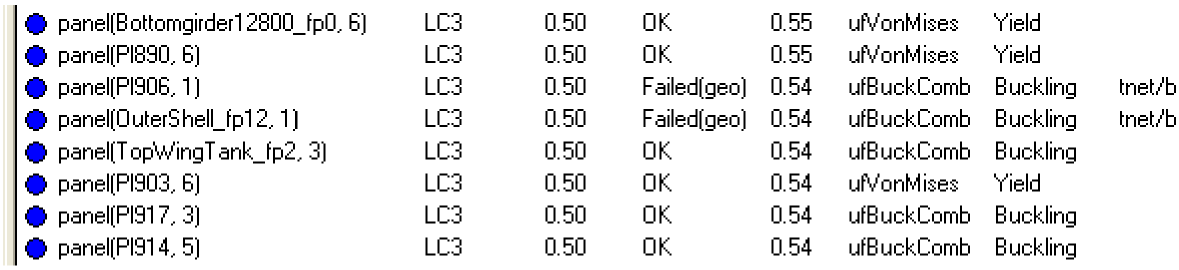
The code check is executed by using the command Execute Code Check.

Like for the *Generate Code Check Loads*, the *Execute Code Check*, can be carried out for all the runs at once or for only one run.

The illustration to the right shows how to execute code check for all runs at once.



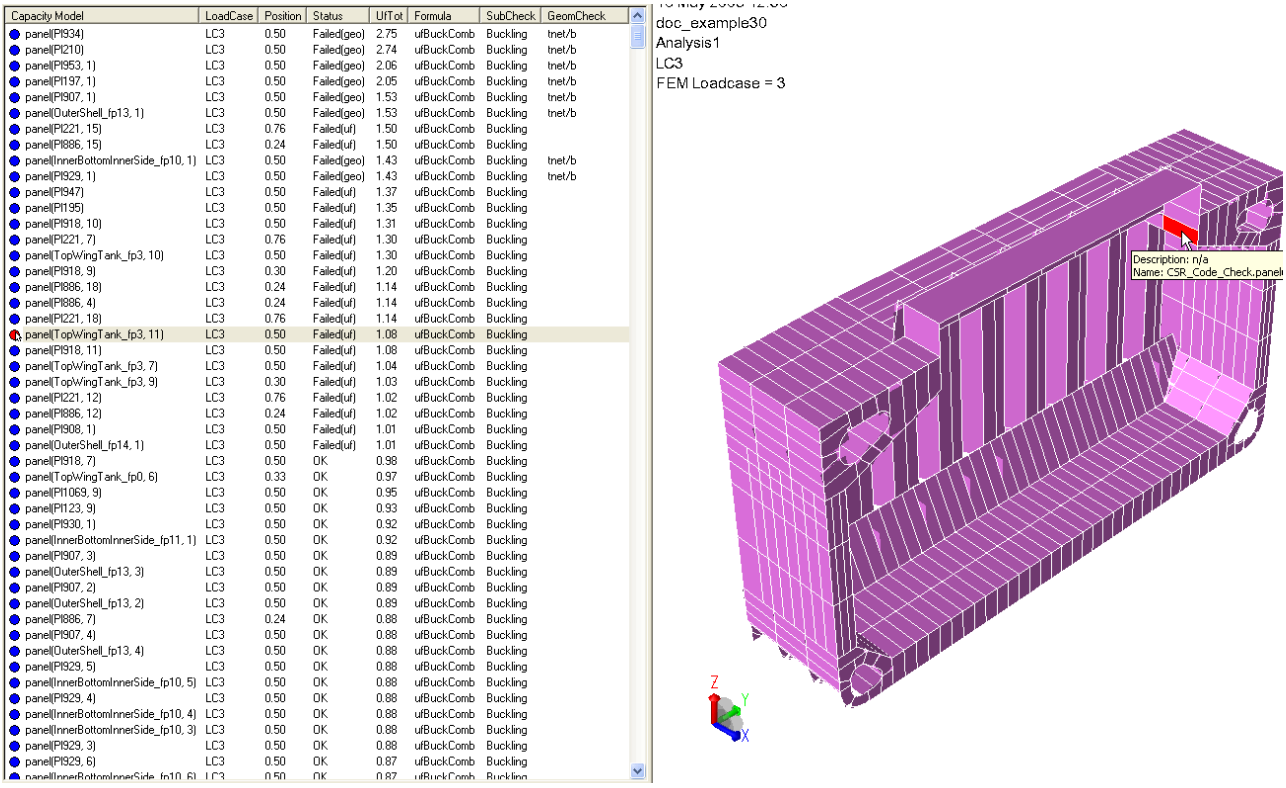
When the code check has been performed the browser includes results from the code check. The example below shows a typical browser view for a panel check.



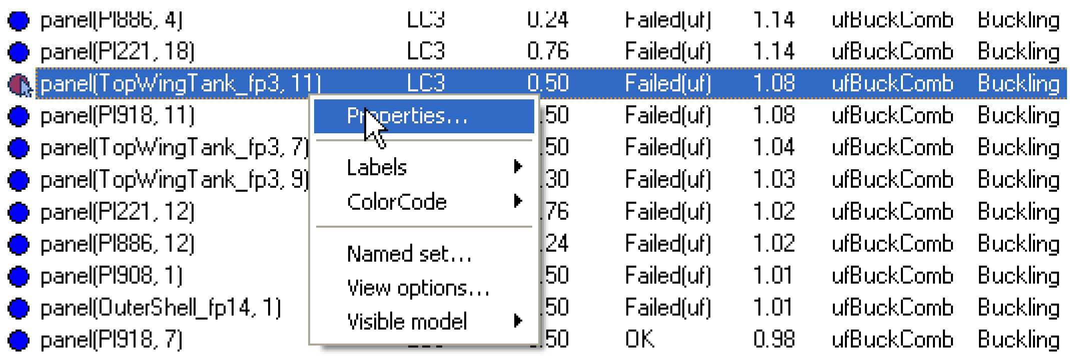
panel(Bottomgirder12800_fp0, 6)	LC3	0.50	OK	0.55	ufVonMises	Yield	
panel(Pl890, 6)	LC3	0.50	OK	0.55	ufVonMises	Yield	
panel(Pl906, 1)	LC3	0.50	Failed(geo)	0.54	ufBuckComb	Buckling	tnet/b
panel(OuterShell_fp12, 1)	LC3	0.50	Failed(geo)	0.54	ufBuckComb	Buckling	tnet/b
panel(TopWingTank_fp2, 3)	LC3	0.50	OK	0.54	ufBuckComb	Buckling	
panel(Pl903, 6)	LC3	0.50	OK	0.54	ufVonMises	Yield	
panel(Pl917, 3)	LC3	0.50	OK	0.54	ufBuckComb	Buckling	
panel(Pl914, 5)	LC3	0.50	OK	0.54	ufBuckComb	Buckling	

3.5 Local code checking parameters

This Section describes how to modify the parameters for individually capacity panels. To do this, you select one ore more panels either in the browser or in the capacity models graphic view. Then click **RMB** and select ***Properties***. You can then modify the code checking parameters as described in the previous Chapter.



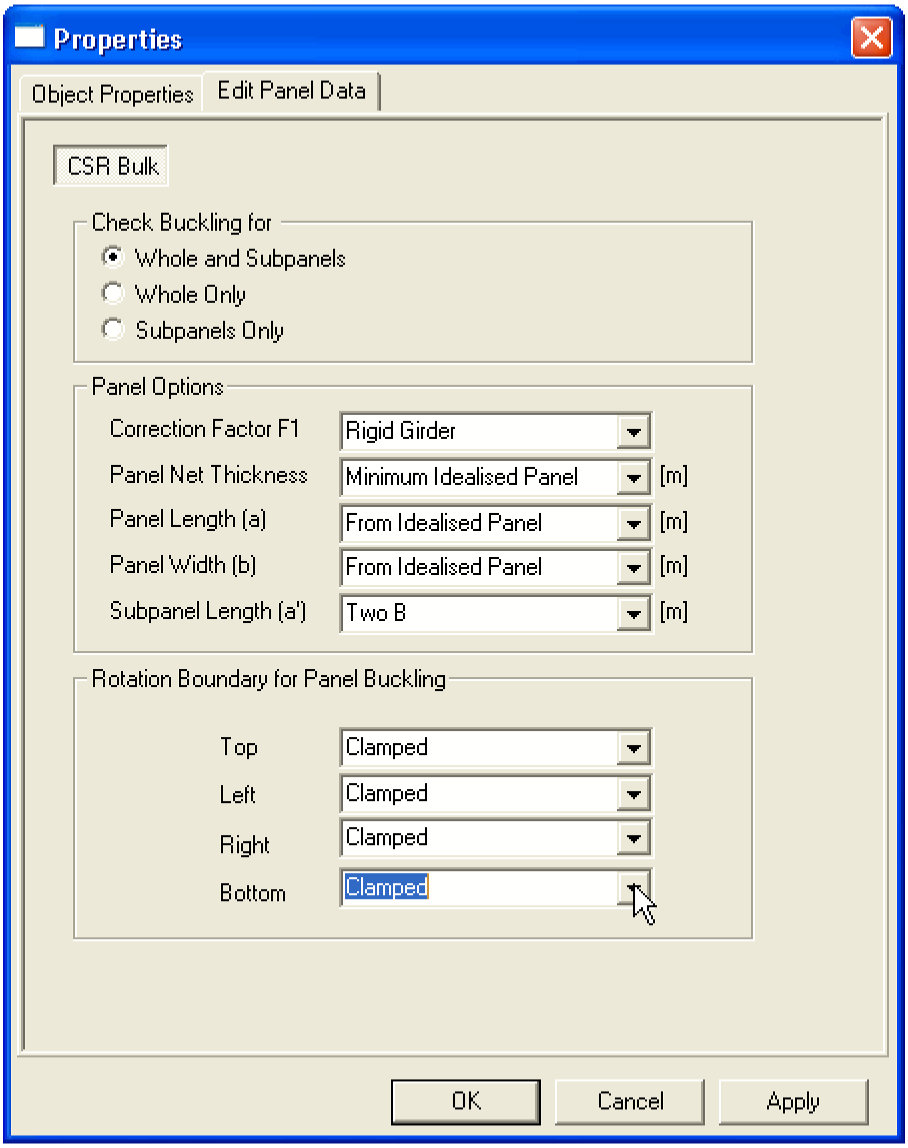
This opens the Properties dialog.

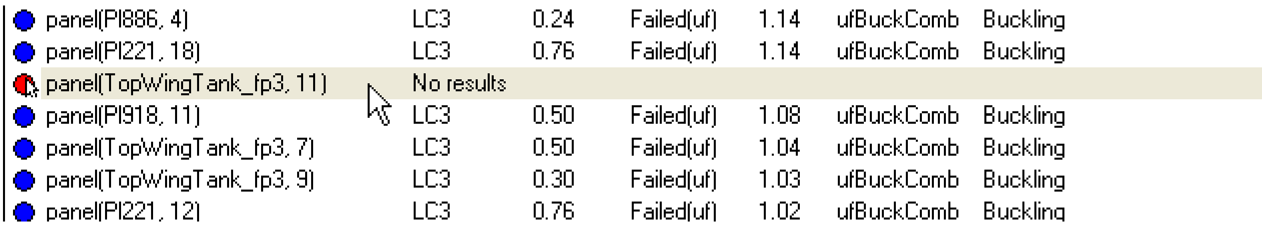


Under the Edit Panel Data tab we can change the panel's properties.

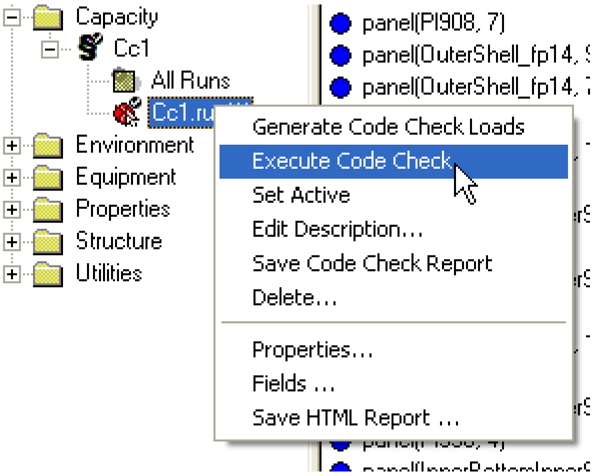
The correction factor F1 is changed from *Sniped* to *Rigid Girder*. The rotation boundaries for panel buckling is changed from *Simply Supported* to *Clamped* for all boundaries.

After closing the Properties dialog by clicking ***OK***, we can observe that there are no results available for the panel.

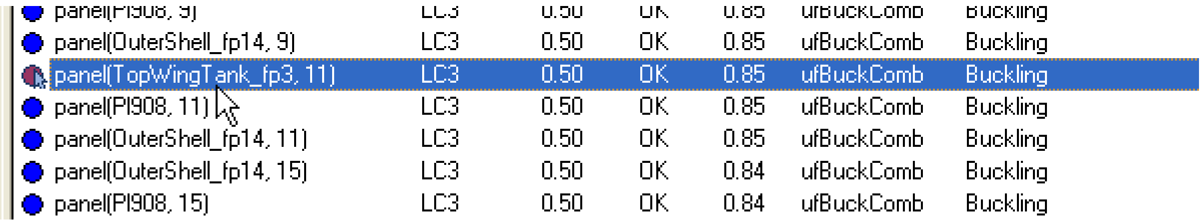




panel(Pl886, 4)	LC3	0.24	Failed(uf)	1.14	ufBuckComb	Buckling
panel(Pl221, 18)	LC3	0.76	Failed(uf)	1.14	ufBuckComb	Buckling
panel(TopWingTank_fp3, 11)	No results					
panel(Pl918, 11)	LC3	0.50	Failed(uf)	1.08	ufBuckComb	Buckling
panel(TopWingTank_fp3, 7)	LC3	0.50	Failed(uf)	1.04	ufBuckComb	Buckling
panel(TopWingTank_fp3, 9)	LC3	0.30	Failed(uf)	1.03	ufBuckComb	Buckling
panel(Pl221, 12)	LC3	0.76	Failed(uf)	1.02	ufBuckComb	Buckling



To see the new result after having made the changes, we have to select ***Execute Code Check*** for our run.

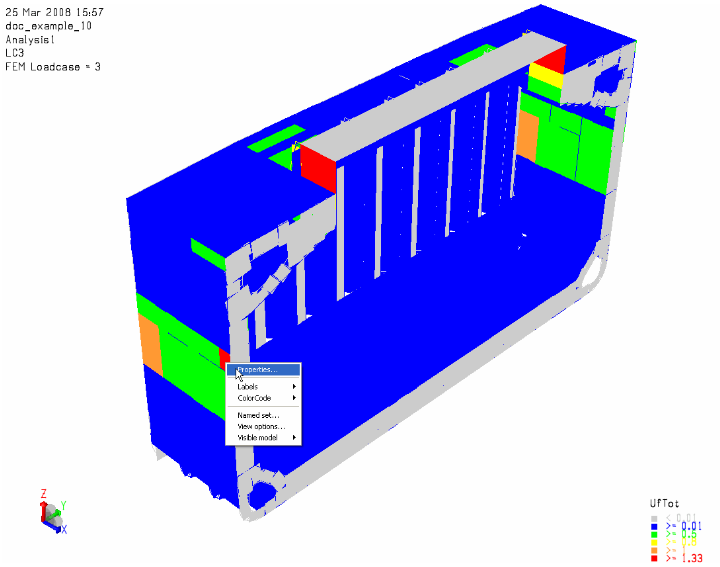


panel(OuterShell_fp14, 9)	LC3	0.50	OK	0.85	ufBuckComb	Buckling
panel(TopWingTank_fp3, 11)	LC3	0.50	OK	0.85	ufBuckComb	Buckling
panel(Pl908, 11)	LC3	0.50	OK	0.85	ufBuckComb	Buckling
panel(OuterShell_fp14, 11)	LC3	0.50	OK	0.85	ufBuckComb	Buckling
panel(OuterShell_fp14, 15)	LC3	0.50	OK	0.84	ufBuckComb	Buckling
panel(Pl908, 15)	LC3	0.50	OK	0.84	ufBuckComb	Buckling

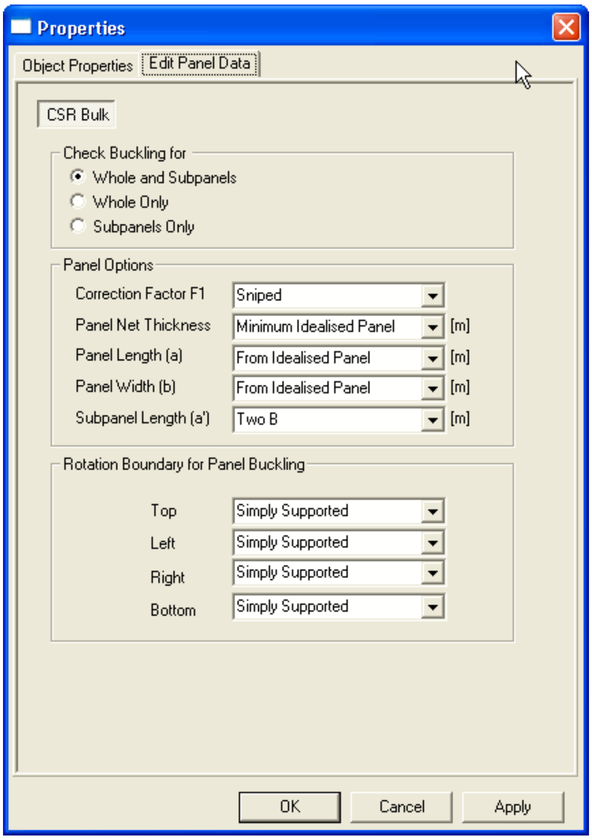
The new result for the panel is now available. Note that Uftot for the panel has changed because of the changes we made in the panel’s properties.

3.6 The properties of a panel

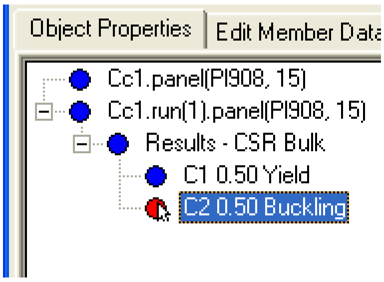
We are watching the capacity model color coded with respect to the Uftot. We select one panel by **LMB**, click **RMB** and select ***Properties***.



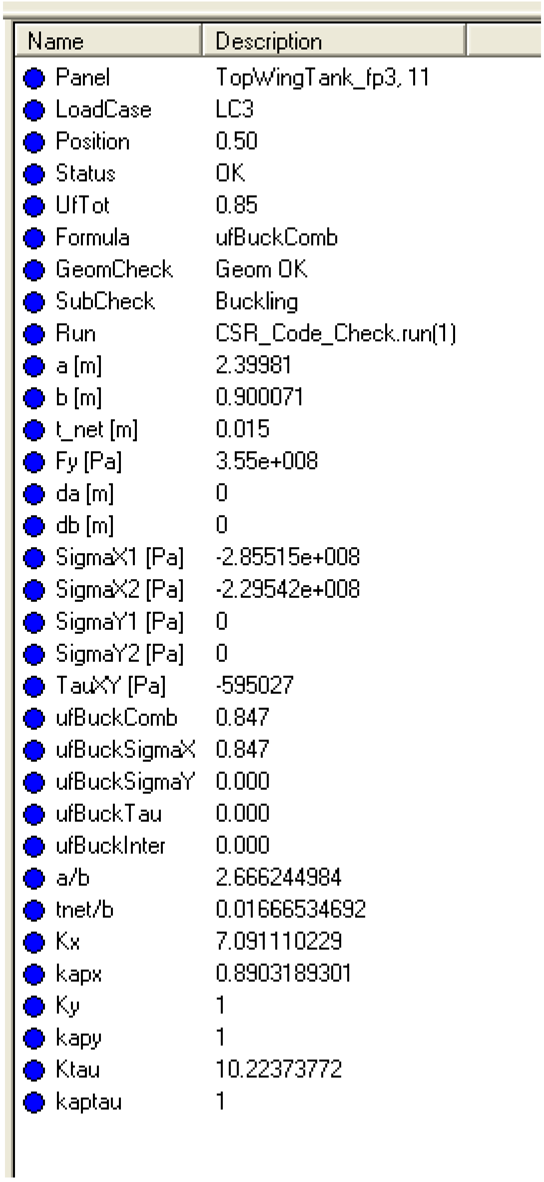
The “Properties” dialog appears and we select ***Object Properties*** to have a look at the properties for the selected panel.



We then select ***Buckling*** to have a look at the buckling results.

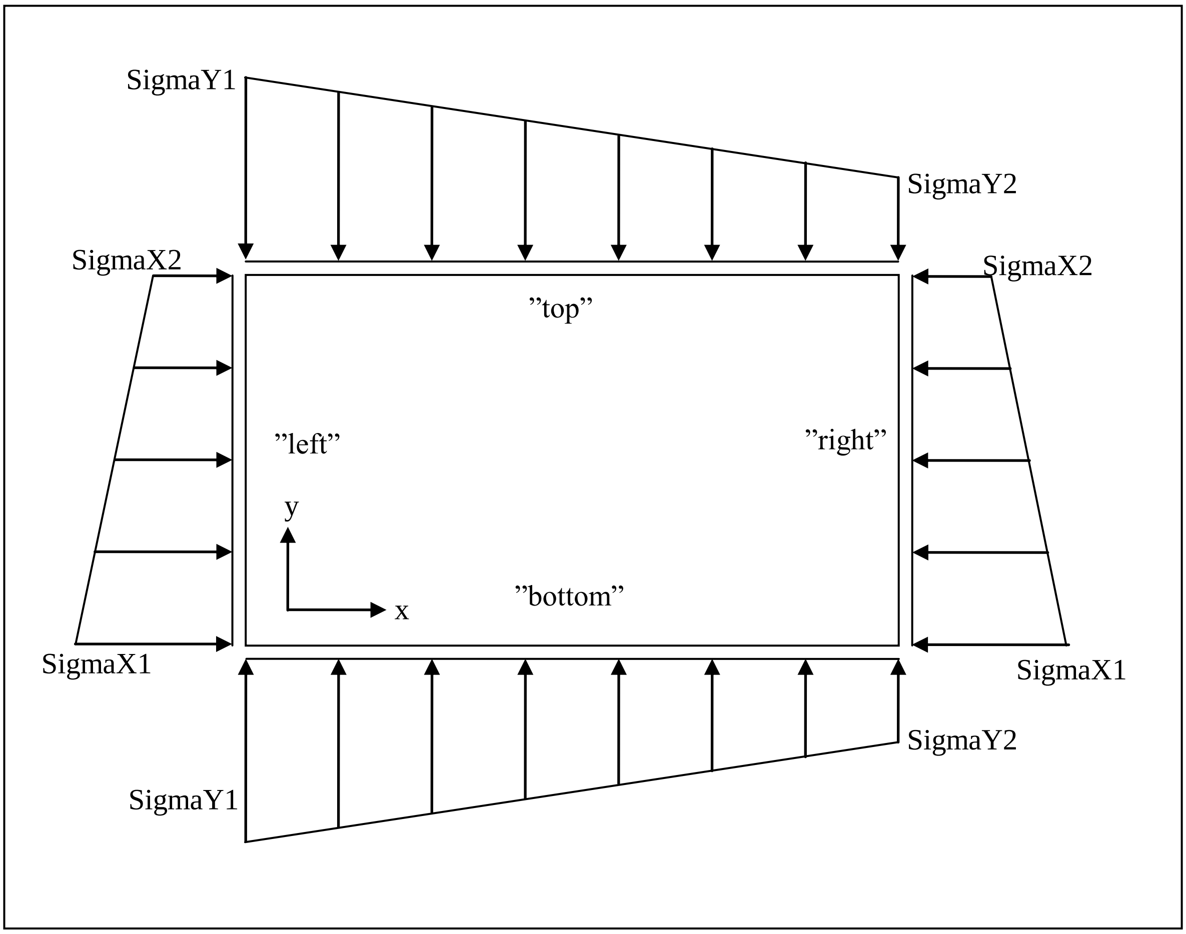


A lot of information concerning the panel is available. The different concepts are explained below.



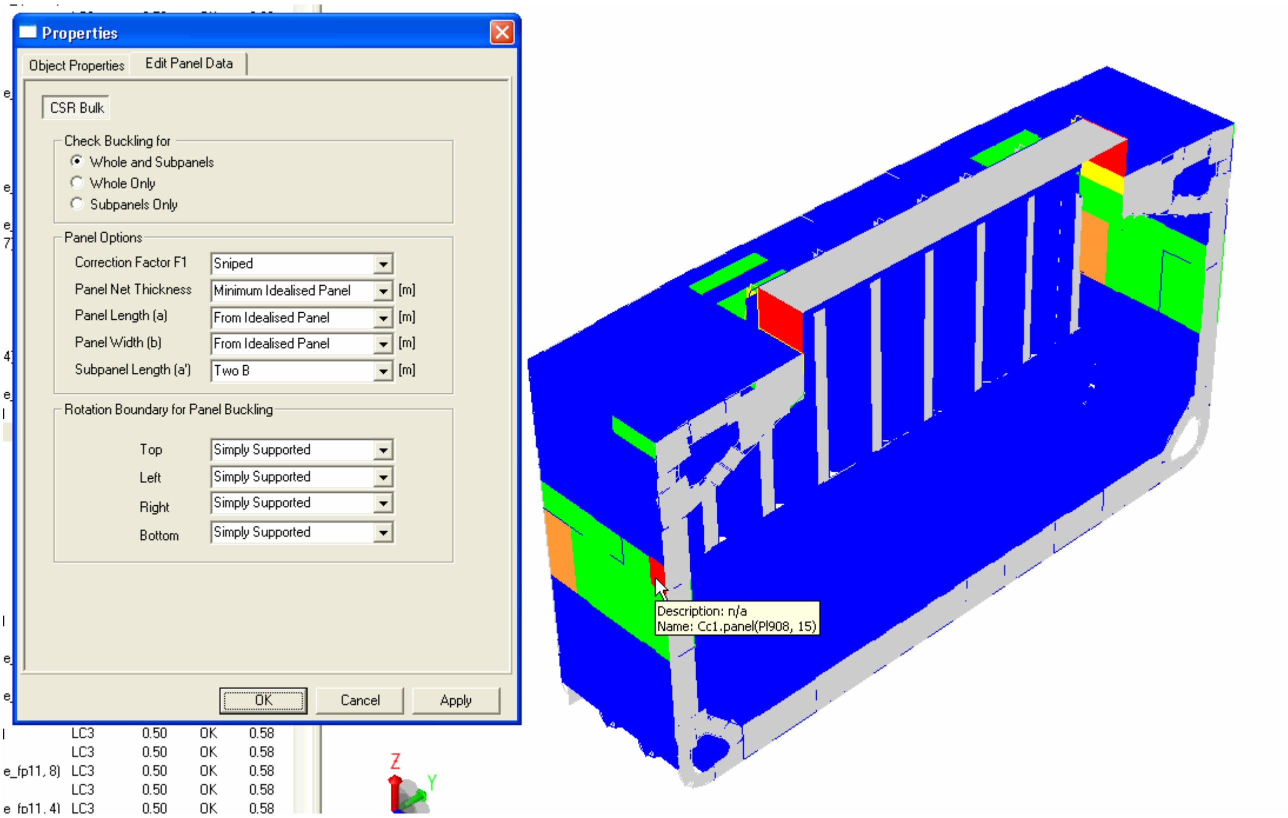
Name	Description
Panel	TopWingTank_fp3, 11
LoadCase	LC3
Position	0.50
Status	OK
UfTot	0.85
Formula	ufBuckComb
GeomCheck	Geom OK
SubCheck	Buckling
Run	CSR_Code_Check.run(1)
a [m]	2.39981
b [m]	0.900071
t_net [m]	0.015
Fy [Pa]	3.55e+008
da [m]	0
db [m]	0
SigmaX1 [Pa]	-2.85515e+008
SigmaX2 [Pa]	-2.29542e+008
SigmaY1 [Pa]	0
SigmaY2 [Pa]	0
TauXY [Pa]	-595027
ufBuckComb	0.847
ufBuckSigmaX	0.847
ufBuckSigmaY	0.000
ufBuckTau	0.000
ufBuckInter	0.000
a/b	2.666244984
tnet/b	0.01666534692
Kx	7.091110229
kapx	0.8903189301
Ky	1
kapy	1
Ktau	10.22373772
kaptau	1

Panel	The name of the panel
Loadcase	The name of the loadcase
Position	Centroid of panel or subpanel. 0.50 is on the middle of whole panel.
Status	Status of a panel is either: *OK*, *Failed (Uf)* or *Failed (geo)*
UfTot	Utilisation factor
Formula	The formula that is governing
GeomCheck	Shows whether the panel's geometry is consistent with regards to the code check
SubCheck	Which subcheck is governing
Run	The name of the run
a	The length of the panel (longest side)
b	The breadth of the panel (shortest side)
t_net	The net thickness of the panel used in capacity check
Fy	Yield strength of material
da	Length of cut out in panel (longest side), if any
db	Breadth of cut out in panel (shortest side), if any
SigmaX1	Stress in X direction along short side*, see illustration*
SigmaX2	Stress in X direction along short side*, see illustration*
SigmaY1	Stress in Y direction along long side*, see illustration*
SigmaY2	Stress in Y direction along long side*, see illustration*
TauXY	Shear stress in panel
ufBuckComb	Combined buckling Uf (i.e. bi-axial stress plus shear)
ufSigmaX	Contribution to Uf from stress in X direction
ufSigmaY	Contribution to Uf from stress in Y direction
ufTau	Contribution to Uf from shear stress
ufInterXY	Contribution to Uf from interaction term
a/b	Aspect ratio a/b
tnet/b	Net thickness / breadth.
Kx	Buckling factor X direction
kapx	Reduction factor X direction
Ky	Buckling factor Y direction
kapy	Reduction factor Y direction
ktau	Buckling factor shear
kaptau	Reduction factor shear


SigmaY1
SigmaY2
SigmaX2
SigmaX2
"top"
"left"
"right"
y
x
"bottom"
SigmaX1
SigmaX1
SigmaY2
SigmaY1

3.7 Edit Panel Data - Description

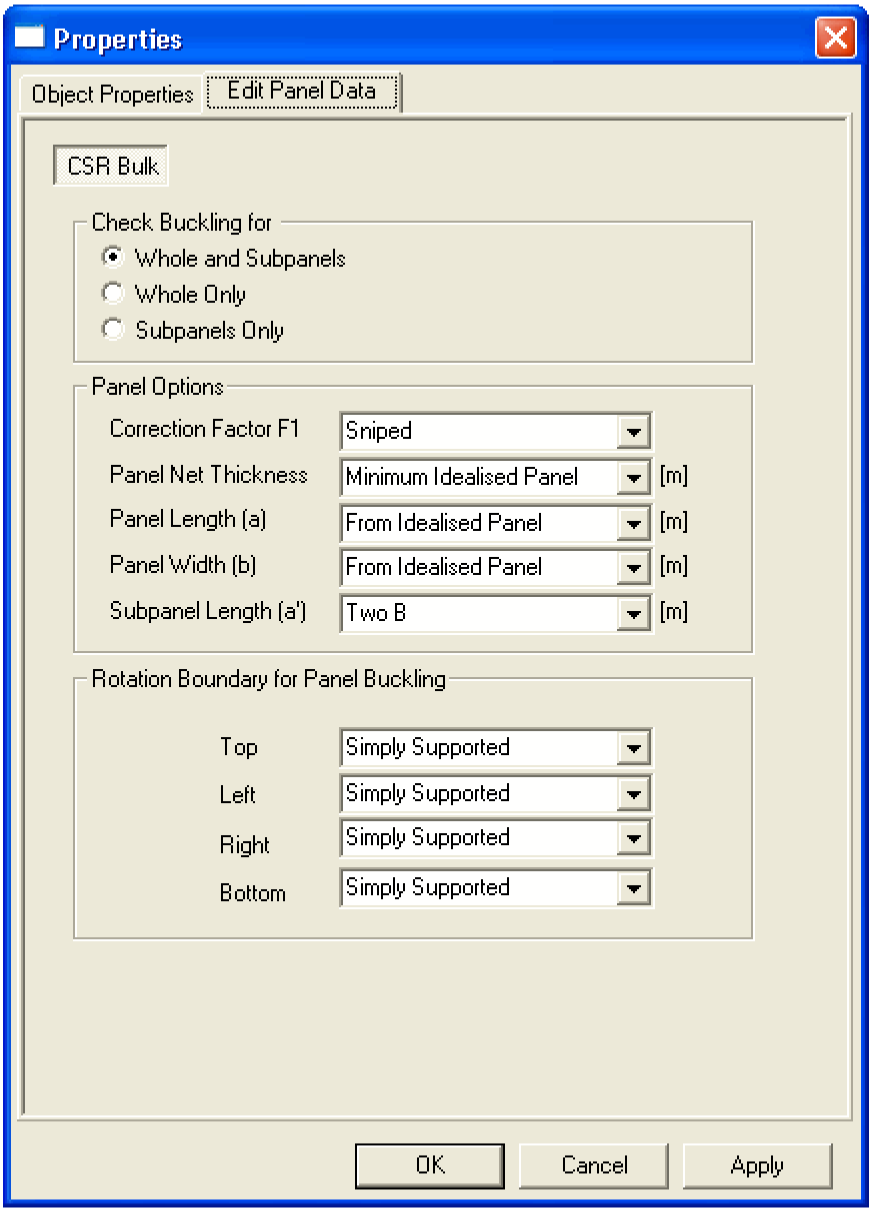
Select a panel, either in the graphics of the capacity view or by using the browser. Click **RMB** and select ***Properties.*** The dialog below shows up. Here you can change the settings for the panel.



3.7.1 Check Buckling for

- Whole and Subpanels
- Whole only
- Subpanels only

These are explained in chapter 2.2.7.

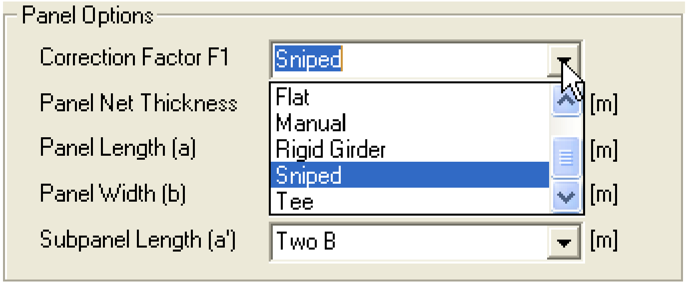


3.7.2 Panel Options

3.7.2.1 Panel Options - Correction Factor F1

The correction factor F1 is described in the CSR Bulk Rules as:

Correction factor for boundary condition of stiffeners on the longer side of elementary plate panels according to Tab 1. If the clamping is unequal on the longitudinal sides of the panel, the minimum value of the appropriate F1 parameter has to be used.



F1 values from the CSR Bulk Rules:

	F1	Edge stiffener
Stiffeners sniped at both ends	1.00	
Guidance values where both ends are effectively connected to adjacent structures	1.05	Flat bar
	1.10	Bulb section
	1.20	Angle and tee-sections
	1.30	Girders of high rigidity (e. g. bottom transverses)

The different F1 values can be chosen in the drop down menu (except the bulb section). Furthermore you can also insert a manual F1 value for your panel.

The default F1 value is *sniped*; F1 = 1.00.

If the panel has different connections to adjacent structure on each side of the panel, you must manually specify the average F1 value.

Example : The panel has a rigid girder on one side and a bulb stiffener on the other side, you must manually specify the average F1 value : F1 = 0.5*(1.3 + 1.1) = 1.2

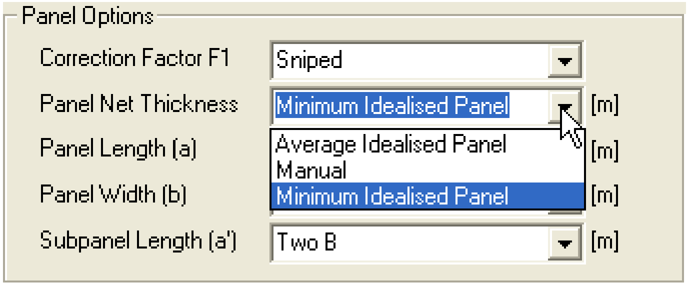
F1 values for corrugated bulkheads

	F1	Edge stiffener
Corrugated bulkheads	1.1	Corr Web - web plate of corrugated bulkhead
	1.1	Corr Flange – flange plate of corrugated bulkhead

Note that normally the value of F1 is not significant for the usage factors for corrugated bulkheads, since the order of magnitude of SigmaY is much smaller than SigmaX.

3.7.2.2 *Panel Options - Panel Net Thickness*

You have three alternatives for setting the net thickness of the panel.



Average Idealised Panel

Using the weighted average net thickness within the panel.

Manual

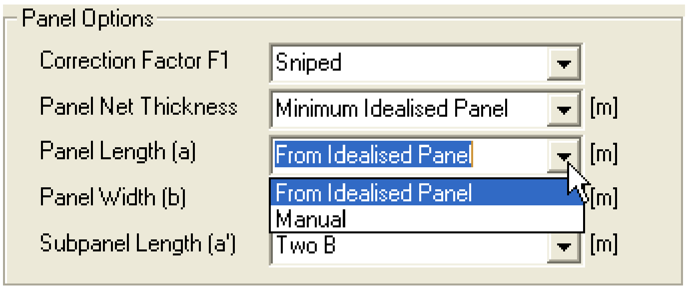
Manually override the **net** thickness

Minimum Idealised Panel (default)

Using the smallest net thickness within the panel.

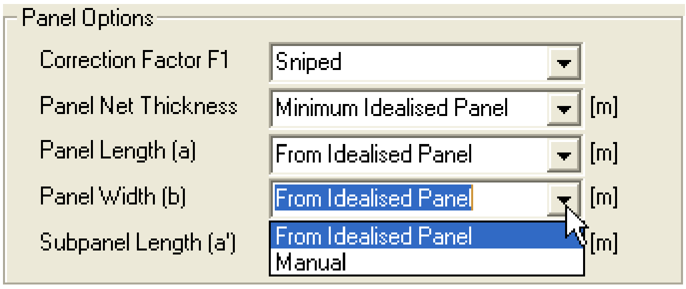
3.7.2.3 *Panel Options - Panel Length (a) and Panel Width (b)*

There are two options:



From Idealised Panel

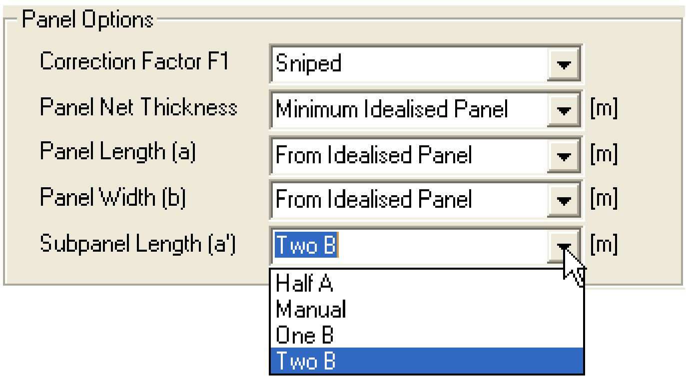
The panel's length and width are decided from the idealised panel.



Manual

After having chosen *Manual* you can fill in your own panel width or length. You can use this option if you are not satisfied with the idealised panel.

3.7.2.4 *Panel Options - Subpanel Length (a')*



If a panel is long compared to its breadth, it will be divided into several subpanels.

In the alternatives here "A" is referring to the length of the "big" panel that surrounds the subpanel. B is referring to the breadth of the "big" panel.

Two B (default)

By default the subpanel length is set to *Two B*, meaning twice the breadth of the “big” panel.

This is what is suitable for buckling of long panels where shear is dominant.

Half A

Use the *Half A* option will be equivalent to inserting a buckling stiffener over the middle of the panel, without this being modelled.

One B

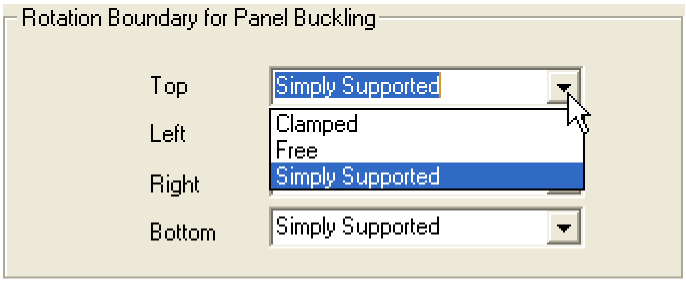
Use the *One B* option for buckling of long panels where axial force is dominant.

Manual

The subpanel length can also be manually overridden.

3.7.2.5 Rotation Boundary for Panel Buckling

The drop down menu alternatives are the same for Top, Left, Right and Bottom. Since they are all identical, only the drop down menu for Top is shown in the illustration.



The alternatives are:

Clamped

Clamping of edges should normally not be specified unless the user is secure that the edge is fully fixed against rotation.

Free

You should set the free edge option if relevant.

Simply Supported (default)

Simply supported is the default option, and it covers most cases.

3.8 Panels and subpanels

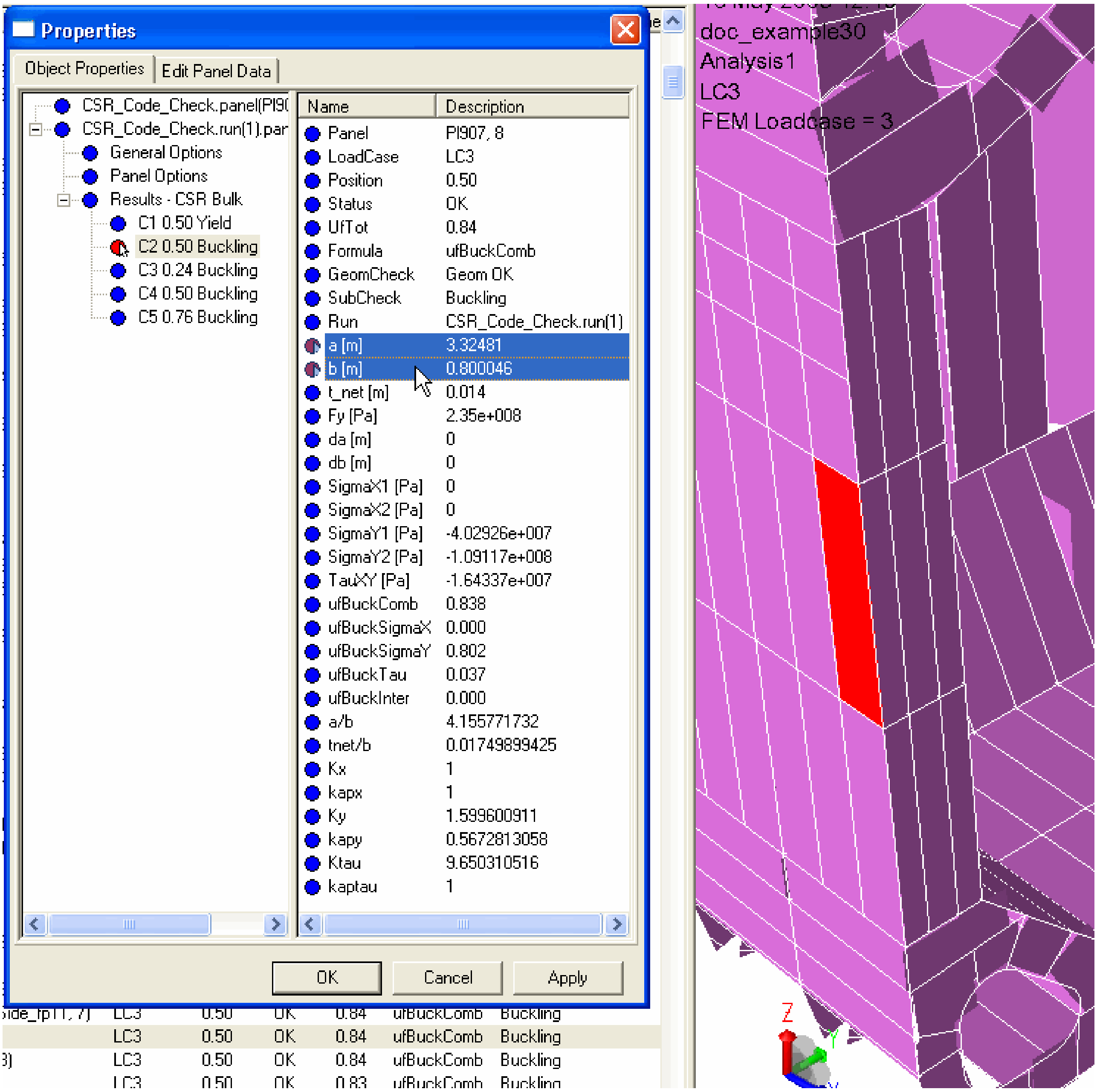
If a panel is long compared to its breadth, subpanels will be created.

Subpanels are created if

a/b >3

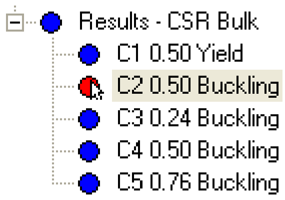
a: the panel's length

b: the panel's breadth



Here we have an example of a panel with length a, 3.32 m and breadth b, 0.8 m. The dimensions of the panel can be seen in the illustration, and the panel is highlighted in the graphics. Note that the subpanels are not visible in the graphics.

The results for the subpanels as well as the main panel can be accessed from the list in the Properties dialog:



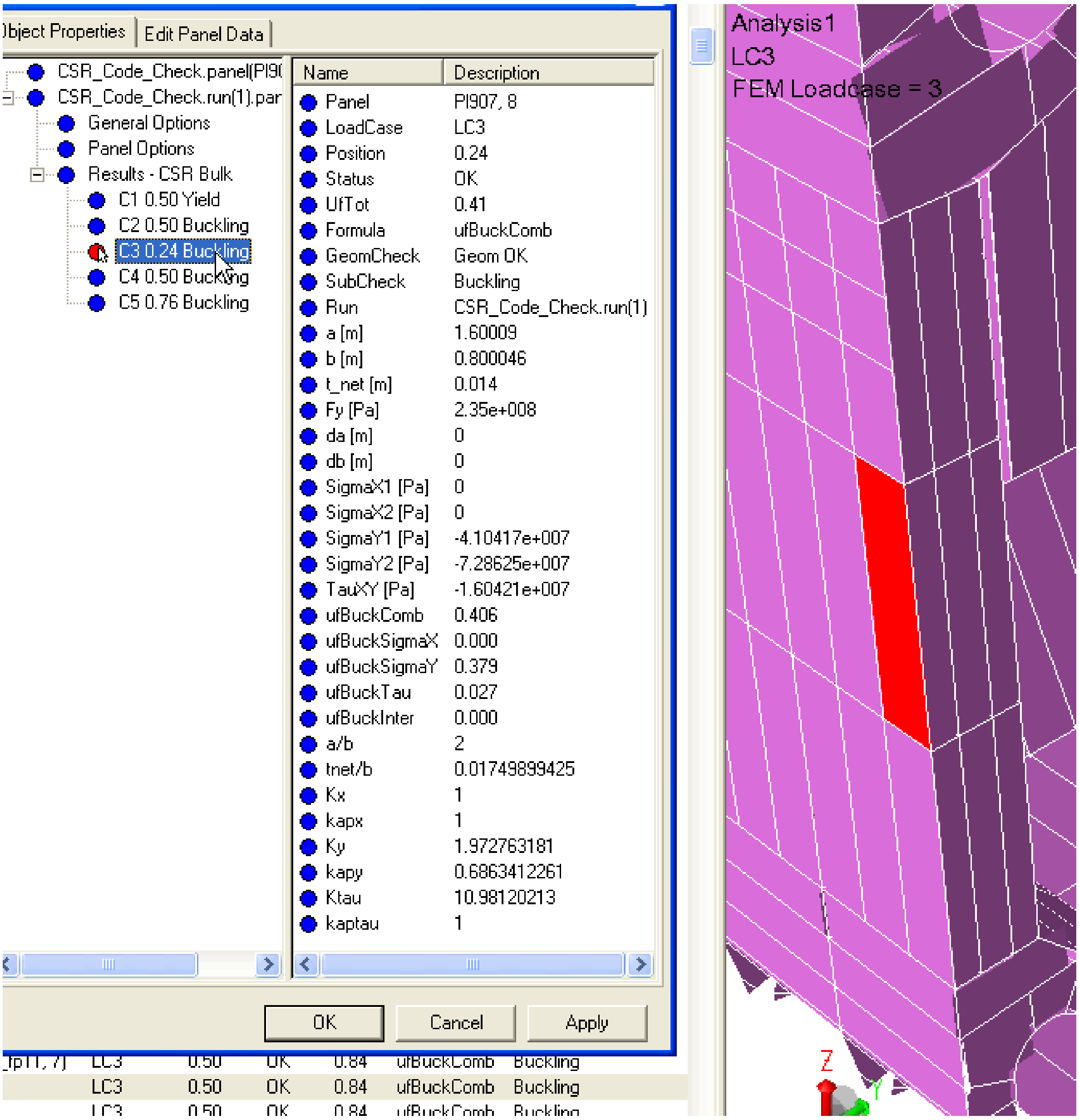
C1 0.50 Yield – This shows the yield results

C2 0.50 Buckling – This shows the results for the entire large panel. The number 0.50 indicates that the results are calculated at the middle of the panel.

C3 0.24 Buckling – This shows the results for the first (out of three) subpanels. The results are calculated at the middle of the subpanel, which in this case is located at 0.24 times the length of the main panel.

C4 0.50 Buckling – Shows the results for the second subpanel.

C5 0.76 Buckling – Shows the results for the third subpanel.



Here we have selected the first subpanel with its centre at 0.24 times the length of the main panel.

a is here the length of the subpanel which is twice the breadth of the main panel by default, it can be changed by the user.

b is here the breadth of the subpanel which is the same as the breadth of the main panel.

If you try to add the lengths of the three subpanels together you see that the total length of the three subpanels are larger than the total length of the main panel. This means that the subpanels overlap each other.

There is never more than five subpanels. This means that for a very narrow main panel there will be a gap between the subpanels. If this is a problem, you can change the length of the subpanels as shown in 3.7.2.4.

Note that the Uftot for this subpanel is smaller than the Uftot for the main panel. In the browser list the largest Uftot is displayed. This is sometimes the Uftot for the main panel and sometimes the Uftot for one of the subpanels.

Below is an example from the browser where the largest Uftot appeared in a subpanel, *Position* is different from 0.50.

Capacity Model	LoadCase	Position	Status	UfTot	Formula	SubCheck	GeomChe
panel(OuterShell_fp13, 3)	LC3	0.50	OK	0.89	ufBuckComb	Buckling	
panel(Pl907, 2)	LC3	0.50	OK	0.89	ufBuckComb	Buckling	
panel(OuterShell_fp13, 2)	LC3	0.50	OK	0.89	ufBuckComb	Buckling	
panel(Pl886, 7)	LC3	0.24	OK	0.88	ufBuckComb	Buckling	
panel(Pl907, 4)	LC3	0.50	OK	0.88	ufBuckComb	Buckling	
panel(OuterShell_fp13, 4)	LC3	0.50	OK	0.88	ufBuckComb	Buckling	
panel(Pl929, 5)	LC3	0.50	OK	0.88	ufBuckComb	Buckling	

3.9 Investigate the results

There are four ways of investigating code checking results:

- From the browser
- Graphically
- From object property
- A report

The three first alternatives are described in this Chapter while Chapter 3.11 “Make a report” shows how to make a default report or how to customize a report.

3.9.1 From the browser

The default view is set up so that it sorts on the highest utilisation factor (UfTot). The results shown are dependent on which load case you have set to active.

1 – If status is Failed(uf), the panel has a usage factor above 1.0 for yield and/or buckling. If buckling gives the highest Uftot, subcheck will be flagged as Buckling.

2 – If status is Failed(geo), the panels fail to satisfy the geometric requirements for the CSR Bulk Panel Buckling code check. The geometry check *tnet/b* failed. This means that the thickness of the panels are too low. To fix this you can increase the thickness in the actual plates, or insert additional stiffeners.

3 – If a panels has a Uftot based on the yield check that is larger than the Uftot based on the buckling check, the Subcheck will be flagged as Yield.

In addition the relative position for the worst code check result for each capacity panel is shown.

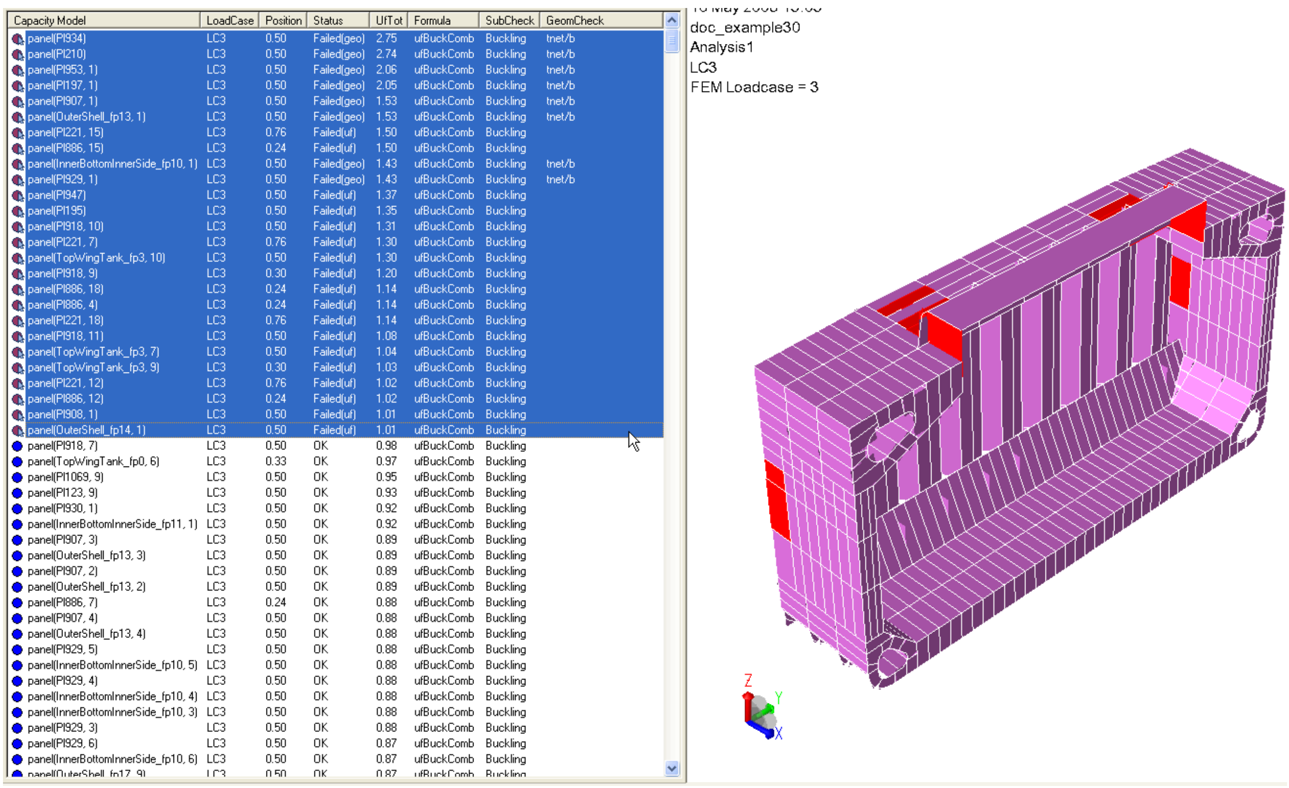
If you specify the option Worst Case (CC) from the loadcase pulldown list, the browser will find the worst condition for each code checking position and report it.

The "Formula" column identifies which formula was used to do the code checking. This is a reference to the relevant formula as listed in the various codes of practice supported by GeniE.

The column for "SubCheck" lists which type of code check that has been performed, i.e. a CSR Bulk Panel Buckling check or a CSR Bulk Panel Yield check. In our case, since we have carried out both code checks, it lists which check that gave the highest Uftot.

You may sort each column in the browser by clicking on the column header.

If you select some of the capacity panels with a high utilisation factor it is easy to see graphically where they are located.



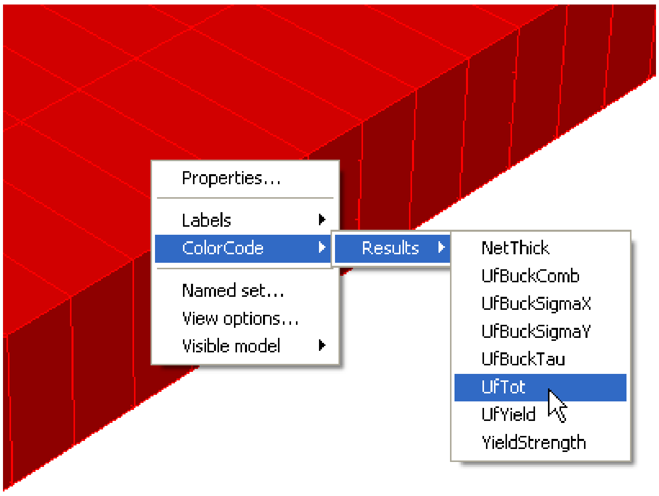
3.9.2 From the graphical window.

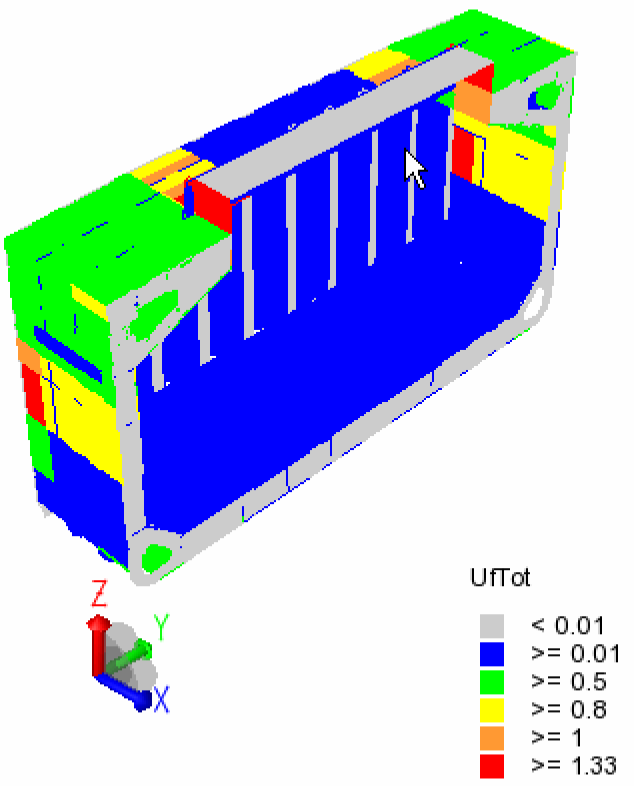
You may select and visualise parts of the capacity model similar to how you do this for other objects like beams, plates, loads etc.

Graphic presentation of results depends on the load case that is selected. You may also select the Worst Case (CC) to display the worst results for all load cases part of the code check.

In the following there are examples on how to present results. Not all are shown herein, but all relevant commands are shown.

Color coding of utilisation factors from code check.

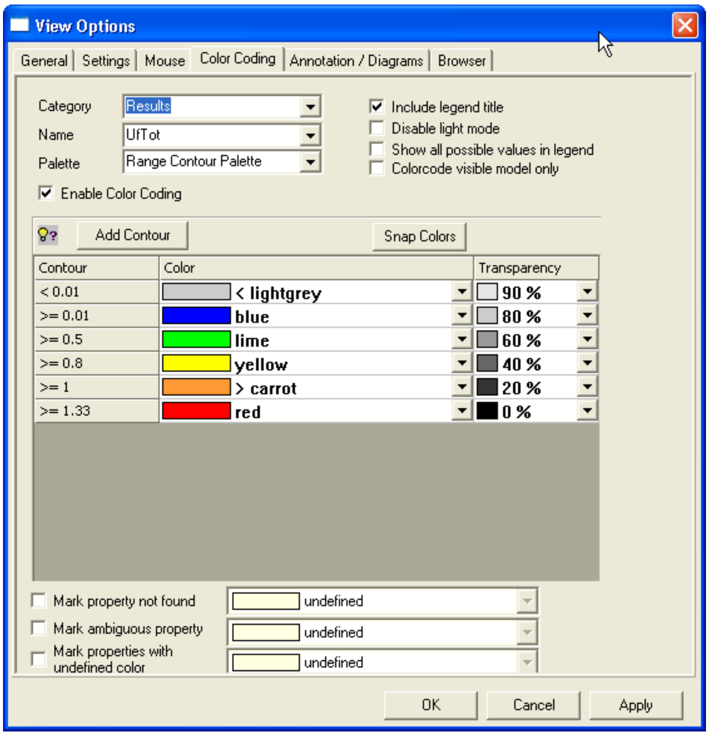




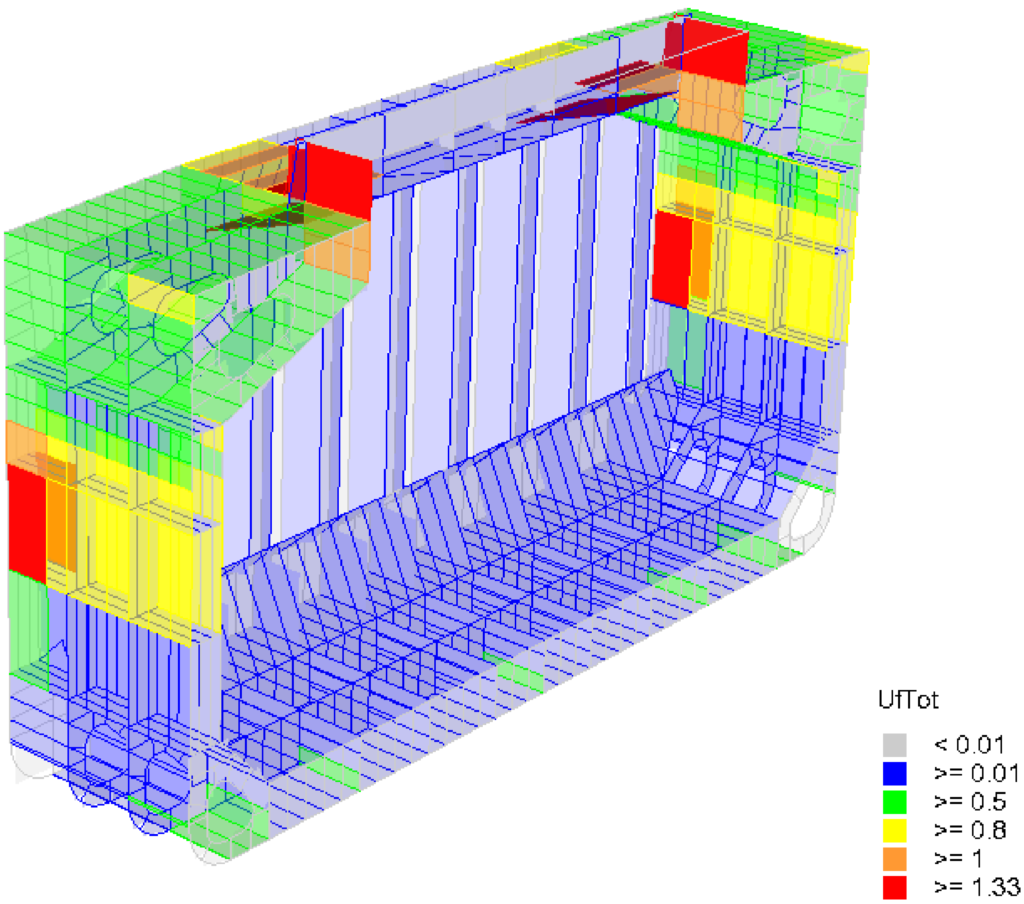
Select all the panels, click **RMB**, select *ColorCode|Result|Uftot.*

As you can see, you also have the option of color coding other information regarding the model. Feel free to try the others as well.

The colour coding is based on default thresholds. You may customise these settings (colours, the threshold values, the number of thresholds as well as the transparency of each colour) from the command *View|Options|Color* Coding.



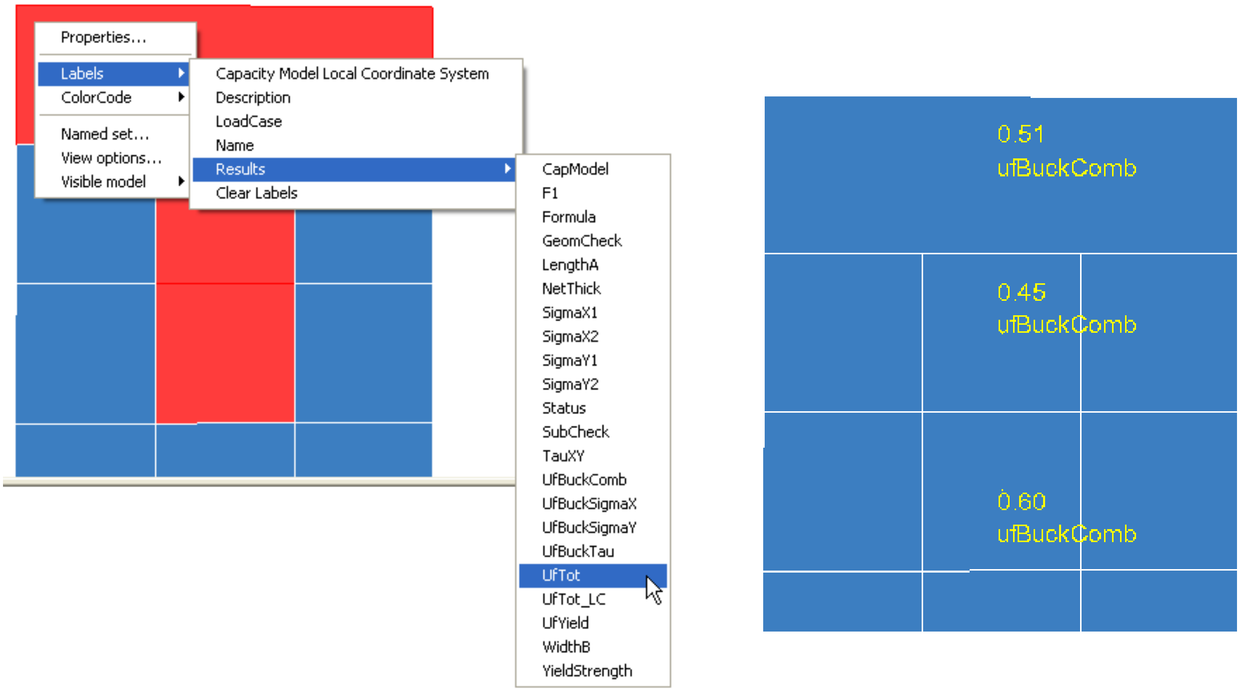
Below is an example of a colour coded model with transparencies set as seen in the illustration to the right. Note that the *Disable light mode* checkbutton also has been unchecked



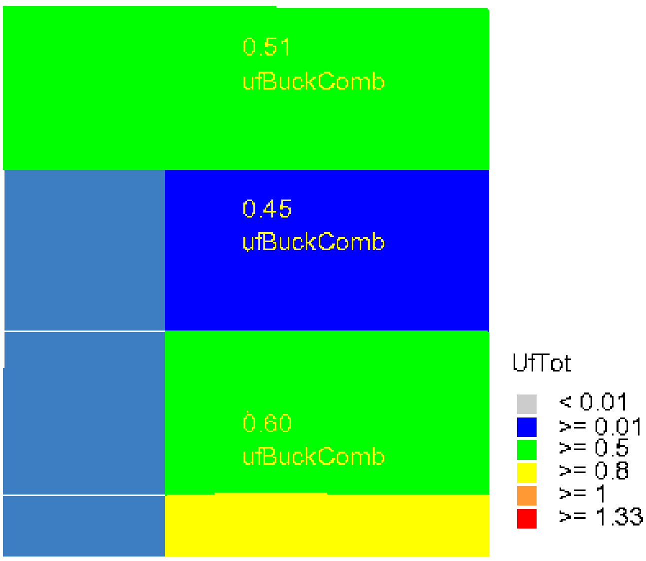
Labelling results from CSR Bulk code check.

To add a label to an object (a capacity panel in this case) you need to select the object and right click to see the available labelling options. You may add labels as shown in the picture below. It is also possible to have several labels on the same object; the example below shows the utilisation factor as well as the formula used during the code checking.

The label UfTot shows the utilisation factor.



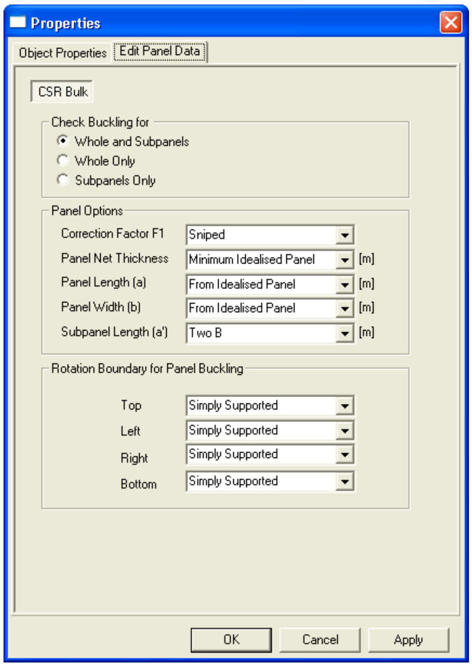
It is also possible to do colour coding of utilisation factors and add labels. In the example below, colour coding has been added.



3.9.3 From object property

The previous ways of documenting the results are primarily meant to give you an overview on whether the code checks are within satisfactory levels or not. If a panel fails it is necessary to investigate the code checking results more detailed to decide how to strengthen the panel. Detailed code checking results may be found from the object browser or from a printed report. Both options are explained in the following.

It may also be that you need to re-distribute the forces in the structure to improve the code checking results. To understand the moment and force distribution you may use the features for visualising these in a 3 dimensional view (Tools|Analysis|Presentation).

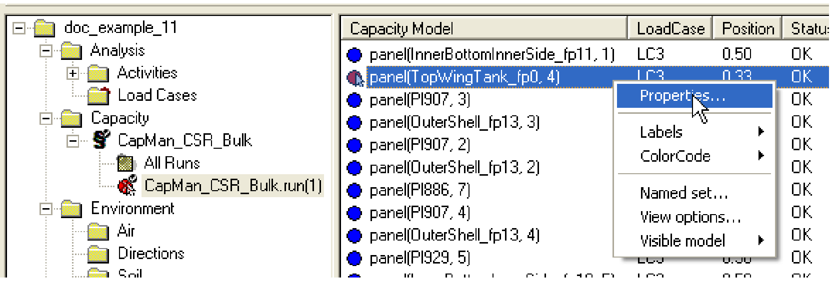


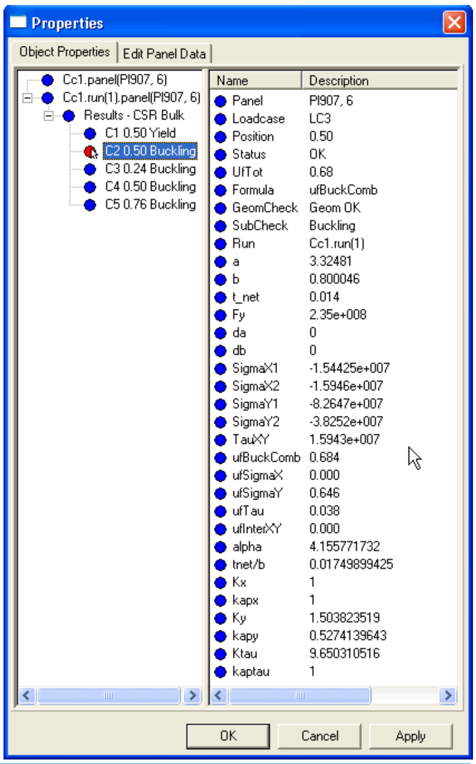
3.9.3.1 Detailed results for panel check.

To access detailed results from the object property you select the panel – either from browser or graphically - you want to investigate, RMB and select *Properties*.

The panel data are listed as shown to the right.

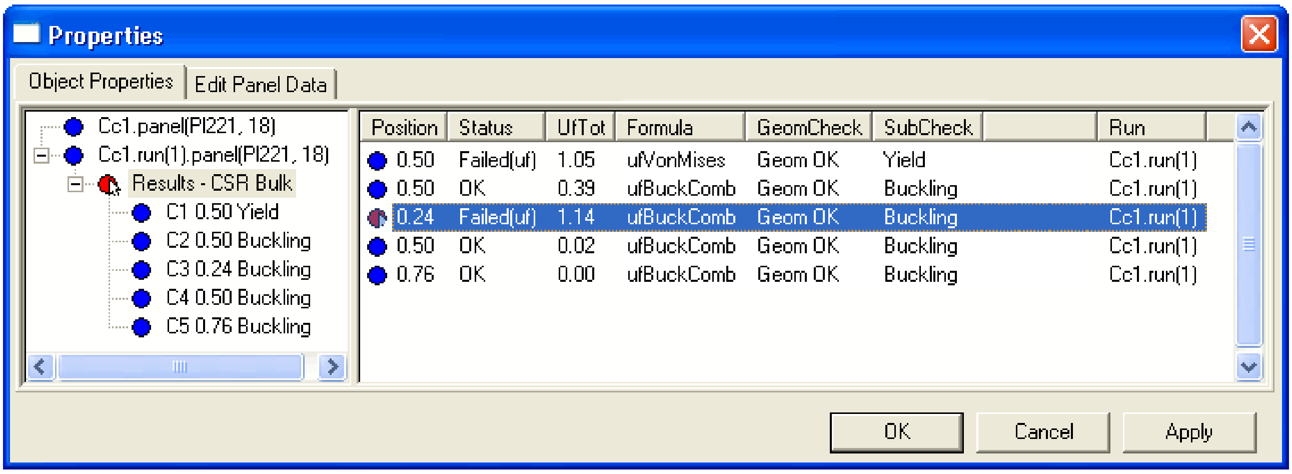
You may also modify panel data for later use.





Panel properties - options.

The Options folder for the Object Properties lists which options have been assigned to the capacity panel. In this case default values have been used. The options to use are described in the previous *Chapter 3.3.2 Define global panel parameters* and *Chapter 3.5 Local code checking parameters*.

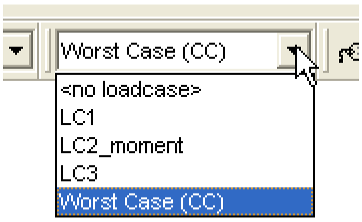


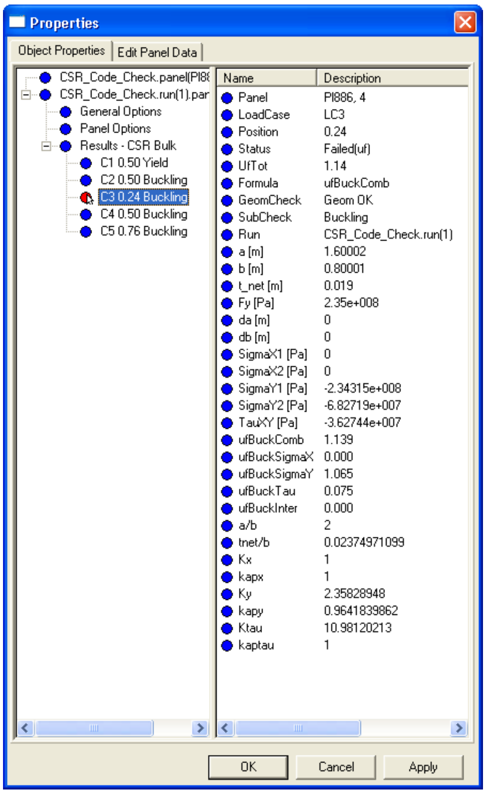
Panel properties – code checking, Uftot.

There are two ways of investigating the code checking forces. The alternative to the right gives the worst forces and moments at each code checking position determined by the program.

The other option is to investigate the forces and moments per position. In this case the worst load case name is also listed.

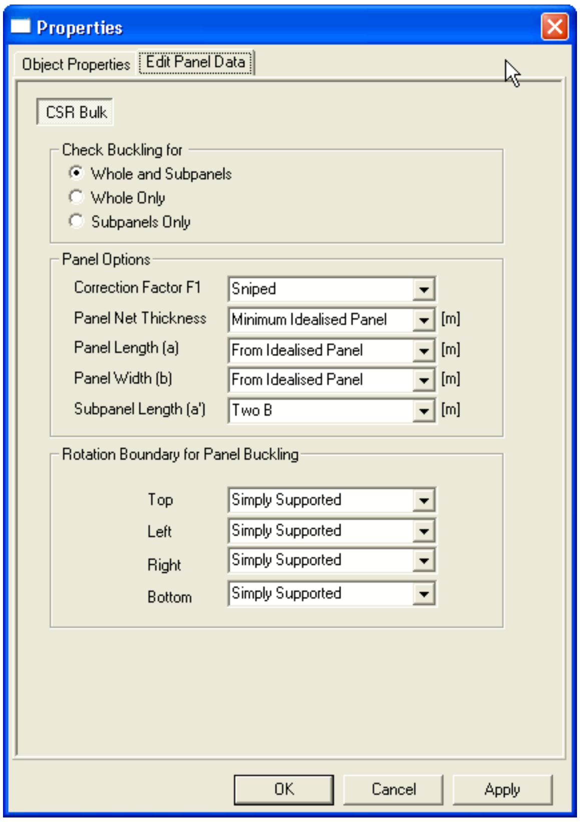
Notice that the load case shown depends on whether you have set a specific load case to active or used the option *Worst Case (CC)*.





3.10 Modify and re-run code check

There are three levels of modifying a code check run:



1) Only *Execute Code Check* needs to be done:

Correction factor F1

Rotation boundaries

These are parameters that belong to the capacity model and a re-run of the code check does not require new code checking forces

2) *Generate Code Check Loads* needs to be done (and *Execute Code Check*):

Check Buckling for:

Whole and Subpanels

Whole Only

Subpanels Only

Panel Options:

Panel net thickness

Panel length

Panel Width

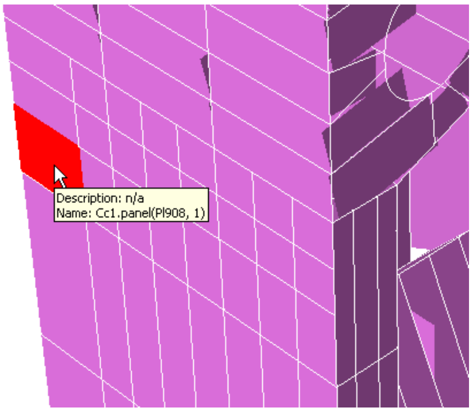
Subpanel Length

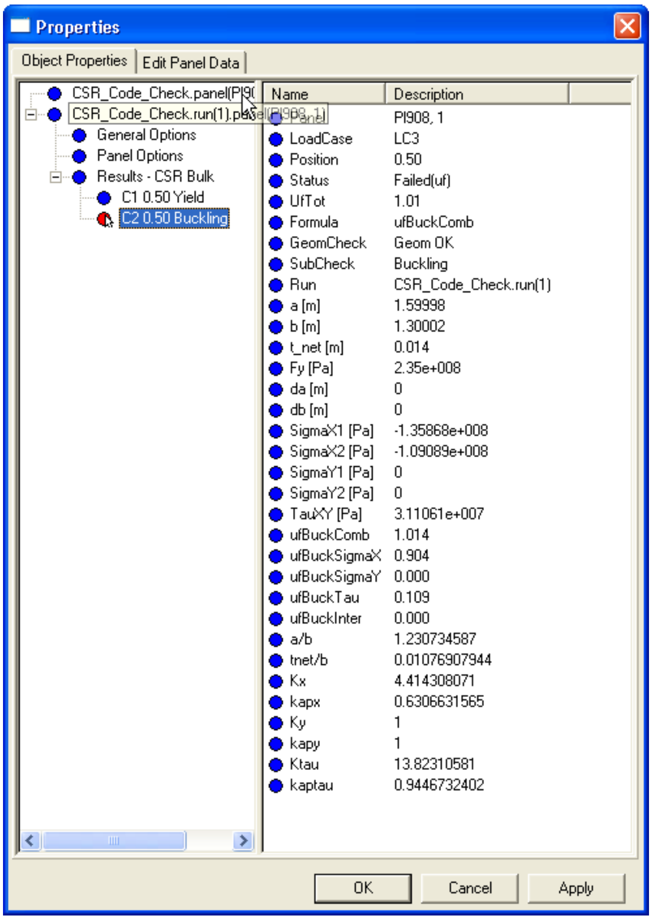
The properties above can be changed without re-running the analysis. Please notice that the code checking result is based on a non-consistent stiffness or load matrix.

3) Entire Analysis needs to be re-run (and *Generate Code Check Loads* and *Execute Code Check*):

Change thickness or material properties or adding/removing additional structural panels using a consistent and updated load and stiffness matrix. This option requires a full re-run of the analysis and the code check steps.

Each of these options are described in the following using the reference model, focus is on panel(Pl908,1). The Uftot for the panel is 1.01.



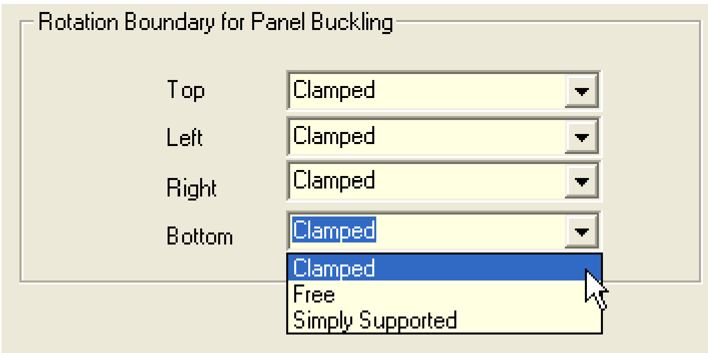


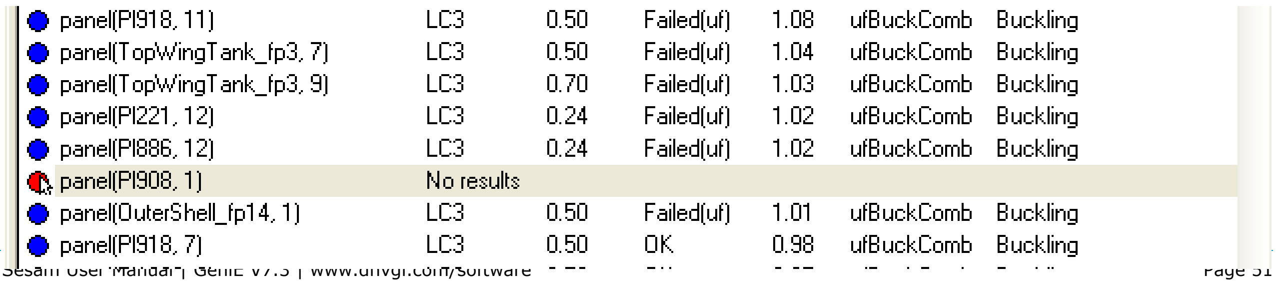
3.10.1 Change code checking parameters only (1)

Modify the rotation boundaries of panel(Pl908,1) from browser or GUI (select the panel, RMB and choose *Properties*).

Change all the rotation boundaries from simply supported to clamped.

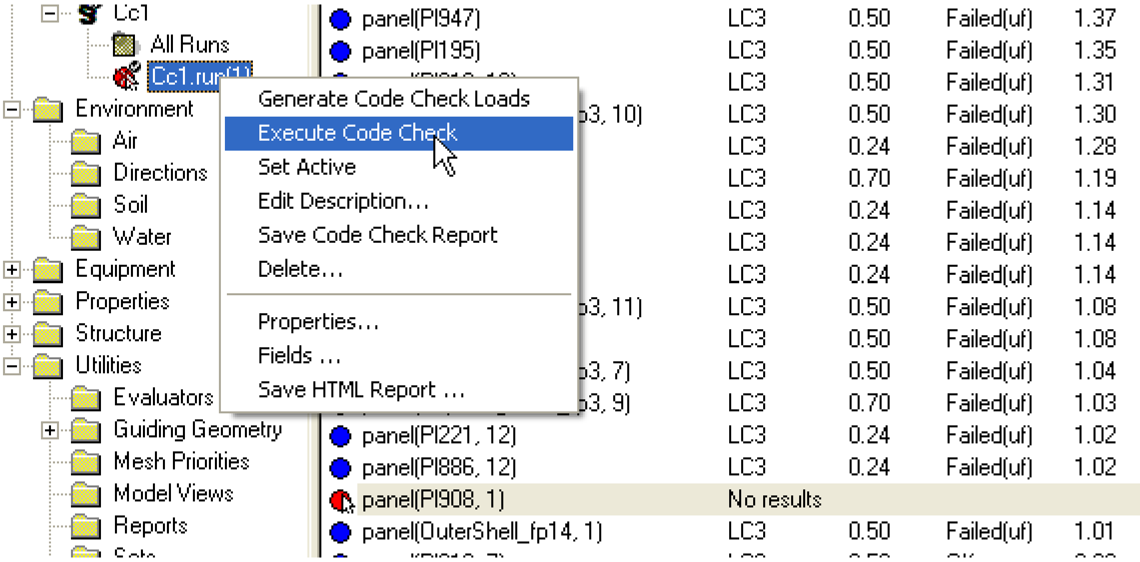
The browser is now updated since panel(Pl908,1) has no results since the panel data has been modified.





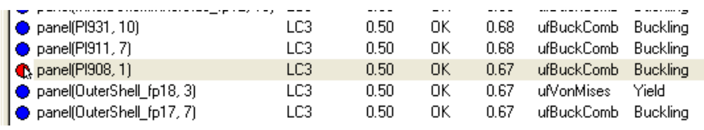
panel(Pl918, 11)	LC3	0.50	Failed(uf)	1.08	ufBuckComb	Buckling
panel(TopWingTank_fp3, 7)	LC3	0.50	Failed(uf)	1.04	ufBuckComb	Buckling
panel(TopWingTank_fp3, 9)	LC3	0.70	Failed(uf)	1.03	ufBuckComb	Buckling
panel(Pl221, 12)	LC3	0.24	Failed(uf)	1.02	ufBuckComb	Buckling
panel(Pl886, 12)	LC3	0.24	Failed(uf)	1.02	ufBuckComb	Buckling
panel(Pl908, 1)	No results					
panel(OuterShell_fp14, 1)	LC3	0.50	Failed(uf)	1.01	ufBuckComb	Buckling
panel(Pl918, 7)	LC3	0.50	OK	0.98	ufBuckComb	Buckling

To generate new code check results you need to re-run the code check. Select the actual run, RMB and choose *Execute Code Check*.



New results are now computed and the may be investigated as explained in the previous Chapter.

As can be seen the utilisation factor has changed from 1.01 to 0.67.



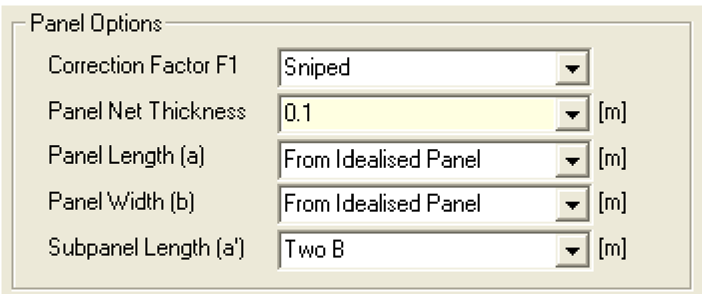
3.10.2 Modify structural data without re-running analysis (2)

When you do modifications to the structural model (typically section or material data) without re-running the whole analysis it is required to re-generate the capacity panels and code checking forces prior to executing the code check.

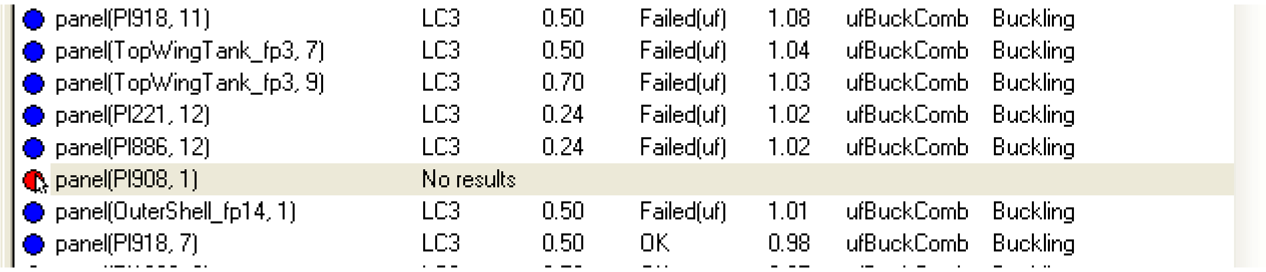
Notice also that if you have added local modifications to a capacity panel these need to be re-applied. All global settings are kept.

The following example shows how to change the panel net thickness and compute a new utilisation factor without re-running analysis. We are starting with our panel(Pl908,1) again. The Uftot for the panel is 1.01.

We right click the panel and select *Properties*. We change *Panel Net Thickness* to Manual and insert the value 0.1.

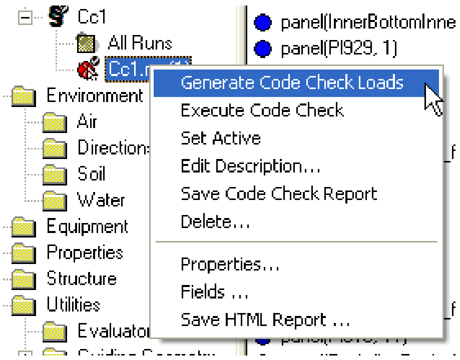


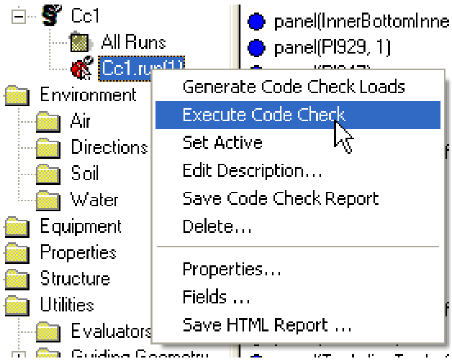
The browser is now updated since panel(Pl908,1) has no results since the panel data has been modified.



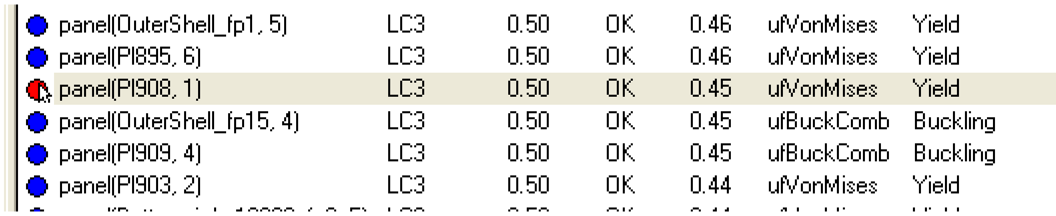
panel(Pl918, 11)	LC3	0.50	Failed(uf)	1.08	ufBuckComb	Buckling
panel(TopWingTank_fp3, 7)	LC3	0.50	Failed(uf)	1.04	ufBuckComb	Buckling
panel(TopWingTank_fp3, 9)	LC3	0.70	Failed(uf)	1.03	ufBuckComb	Buckling
panel(Pl221, 12)	LC3	0.24	Failed(uf)	1.02	ufBuckComb	Buckling
panel(Pl886, 12)	LC3	0.24	Failed(uf)	1.02	ufBuckComb	Buckling
panel(Pl908, 1)	No results					
panel(OuterShell_fp14, 1)	LC3	0.50	Failed(uf)	1.01	ufBuckComb	Buckling
panel(Pl918, 7)	LC3	0.50	OK	0.98	ufBuckComb	Buckling

We select *Generate Code Check Loads* and then *Execute Code Check*:





As can be seen the utilisation factor has changed from 1.01 to 0.45.



panel(OuterShell_fp1, 5)	LC3	0.50	OK	0.46	ufVonMises	Yield
panel(Pl895, 6)	LC3	0.50	OK	0.46	ufVonMises	Yield
panel(Pl908, 1)	LC3	0.50	OK	0.45	ufVonMises	Yield
panel(OuterShell_fp15, 4)	LC3	0.50	OK	0.45	ufBuckComb	Buckling
panel(Pl909, 4)	LC3	0.50	OK	0.45	ufBuckComb	Buckling
panel(Pl903, 2)	LC3	0.50	OK	0.44	ufVonMises	Yield

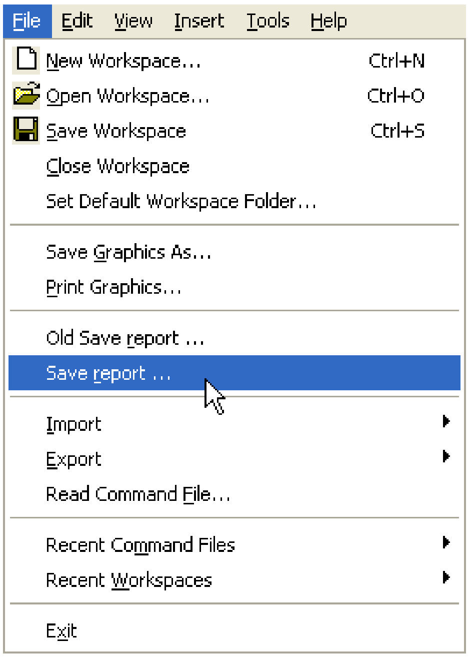
3.10.3 Modify structural data and re-run analysis (3)

When you re-run analysis it is necessary to

- Create new panels of the capacity manager.
    - Global settings are kept, but local modifications must be re-applied
- Compute new code checking forces
- Run the code check

To do this you run analysis (ALT+D) and follow the steps as explained in the previous Section. New results may now be assessed.

3.11 Make a report



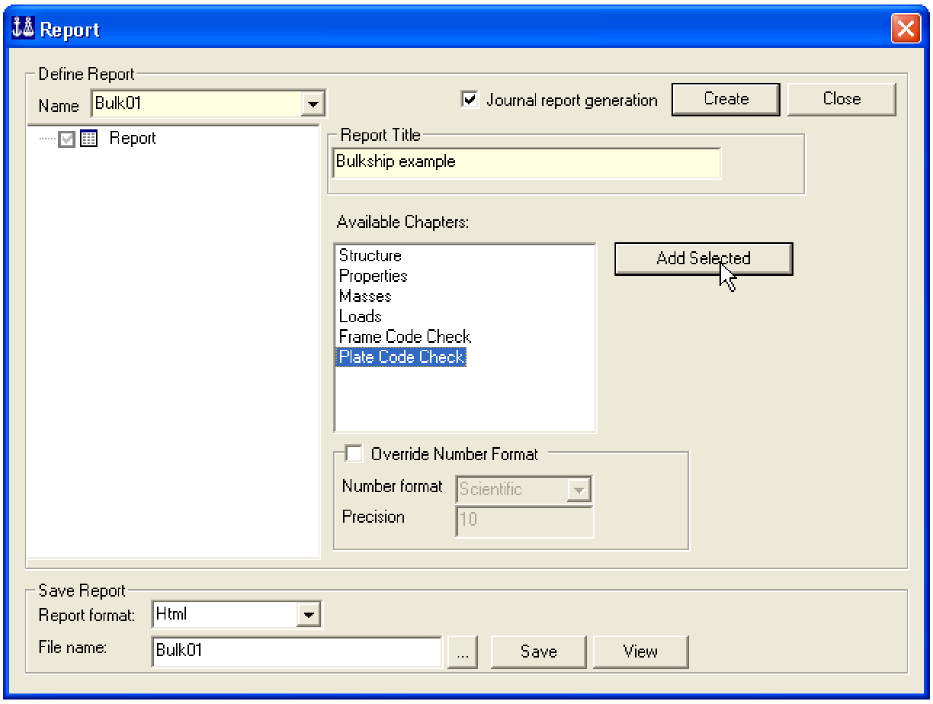
A customised report may be generated from the ***File|Save report***. Please notice that the report functionality in versions prior to GeniE version v3.4-27 is still available from the command ***File|Old Save report***.

The **File|Save report** allows you to specify the content of your report as well as to decide the levels of detail in of your report.

In the following is given an example on how to make a code checking report.

To make a report of the plate code check, you select *Plate Code Check* under *Available Chapters* and click *Add Selected*.

The report name, title and file name can be changed. By default the file name is the same as the report name.

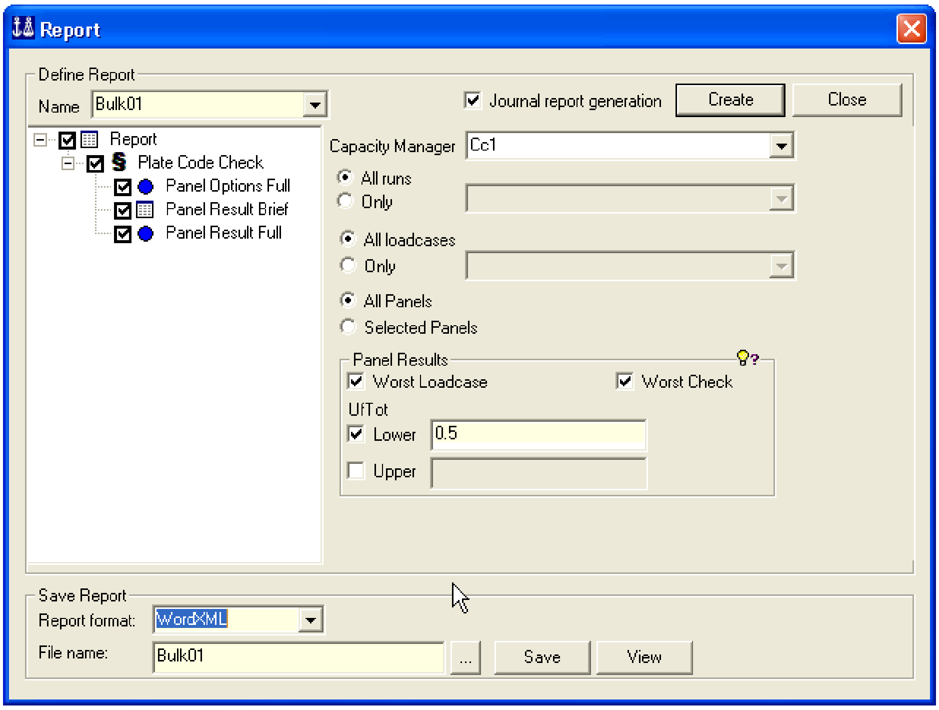


The available tables for plate code check are *Panel Options Full*, *Panel Result Brief* and *Panel Result Full*.

We have checked the *Lower* checkbox for Uftot, and typed in the value 0.5.

This means that the report will not list any panels having a Uftot at 0.5 or lower.

We have chosen the report format to be *WordXML*. Other available formats are: *HTML*, *ExcelXML* and *Text*.



After having clicked *Save*, you can have a look at the report by clicking *View*.

Below are examples of what our selected tables might look like in the report.

Panel Options Full:

Cc1.run(1) : Panel Options CSR Bulk

- Sorted by Panel (Ascending)

- Run : Cc1.run(1)
- Only panels with options modified from the Default object(s) will be listed

Panel	Code	CheckWhole	CheckSubpanel	PanelLength	PanelWidth	PanelThickness	SubpanelLength	F1	RB_Top
	RB_Bottom	RB_Left	RB_Right						
Default	CSR Bulk	true	true	From Idealised Panel	From Idealised Panel	Minimum Idealised Panel	Two B	Sniped	Simply Supported
	Simply Supported	Simply Supported	Simply Supported						
Pl918, 11	CSR Bulk	true	true	From Idealised Panel	From Idealised Panel	Minimum Idealised Panel	Two B	Rigid Girder	Simply Supported
	Simply Supported	Simply Supported	Simply Supported						

Here the options are different from the defaults for only one panel, and this panel is listed in addition to the default values.

Panel Result Brief:

Cc1.run(1) : Panel Result Brief

- Sorted by UfTot (Descending)
- Filtered by Limit : (UfTot >= 0.5)
- Run : Cc1.run(1)
- Worst LoadCase per Panel
- All SubChecks per Panel
- Worst of Whole Panel and SubPanel Checks per Panel

Panel	Loadcase	Position	Status	UfTot	Formula	GeomCheck	SubCheck	Run
Pl934	LC3	0.50	Failed(geo)	2.75	ufTotalFEM	tnet/b	CSR Bulk Panel Buckling	Cc1.run(1)
Pl210	LC3	0.50	Failed(geo)	2.74	ufTotalFEM	tnet/b	CSR Bulk Panel Buckling	Cc1.run(1)
Pl953, 1	LC3	0.50	Failed(geo)	2.07	ufTotalFEM	tnet/b	CSR Bulk Panel Buckling	Cc1.run(1)
Pl197, 1	LC3	0.50	Failed(geo)	2.05	ufTotalFEM	tnet/b	CSR Bulk Panel Buckling	Cc1.run(1)
Pl907, 1	LC3	0.50	Failed(geo)	1.53	ufTotalFEM	tnet/b	CSR Bulk Panel Buckling	Cc1.run(1)

Panel Result Full:

1.1.2 CSR_Code_Check.run(1) : Panel Result Full

1.1.2.1 CSR_Code_Check.run(1) : CSR Bulk Panel Buckling Result

CSR_Code_Check.run(1) : CSR Bulk Panel Buckling Result

- Sorted by Panel (Ascending)
- Then sorted by LoadCase (Ascending)
- Filtered by Limit : (UfTot >= 0.5)
- Run : CSR_Code_Check.run(1)
- Worst LoadCase per Panel
- Selected SubCheck per Panel
- Worst of Whole Panel and SubPanel Checks per Panel

Panel	LoadCase	Position	Status	UfTot	Formula	GeomCheck	SubCheck	Run
	a [m]	b [m]	t_net [m]	Fy [Pa]	da [m]	db [m]	SigmaX1 [Pa]	SigmaX2 [Pa]
	SigmaY1 [Pa]	SigmaY2 [Pa]	TauXY [Pa]	ufBuckComb	ufBuckSigmaX	ufBuckSigmaY	ufBuckTau	ufBuckInter
	a/b	tnet/b	Kx	kapx	Ky	kapy	Ktau	kaptau
Bottomgirder0_fp0, 9	LC3	0.76	OK	0.18	ufBuckComb	Geom OK	Buckling	CSR_Code_Check.run(1)
	1.15012	0.575062	0.013	3.15e+008	0	0	1.04901e+008	1.26932e+008
	-1.26361e+007	3.26827e+007	3.28332e+007	0.184	0.135	0.002	0.033	0.015
	2	0.02260624617	1	1	19.21361732	1	10.98120213	1
Bottomgirder9600_fp0, 7	LC3	0.76	OK	0.18	ufBuckComb	Geom OK	Buckling	CSR_Code_Check.run(1)
	1.15012	0.575062	0.011	3.15e+008	0	0	1.03885e+008	1.26119e+008
	-1.24038e+007	3.24593e+007	3.34317e+007	0.183	0.133	0.002	0.034	0.014

3.12 APPLICATION TO CSR BULK

When performing rule check according to CSR Bulk, capacity checks should be performed for the region around the middle hold of the 3 hold model.

It is recommened to split the check into runs for different sets of structure categories :

Then it will be more easy to set the control parameters that differs for different category. This applies to the subpanel types to be checked and to different F1 values.

	Yield check	**Buckling checks to be performed**		
Part of ship		**Whole panel**	**2 b * b subpanels**	**b x b subpanels**
General	X	X	X (if a/b > 3)	
Corr bhd –web	X	X	X	
Corr bhd – flange	X	X		X
Side shell	X	X	X	X

The generic stress algorithm is extended to be used also for Corrugated bulkhead and Side shell.

This check will use interpolated / averaged stresses for the b x b or for the 2b x b subpanel regions.

For the **corrugated bulkheads – web plate**, the shear stress case is most important and the 2b x b subpanel must be checked.

For the **corrugated bulkheads – flange plate**, the axial stress case is most important and the b x b subpanel must be checked.

For the **side shell**, it will be required to set up an extra buckling run with subpanel size b x b, since it is not possible to do the b x b and 2b x b subpanels in the same run.

4 HOW TO DO PANEL CODE CHECKING – CSR TANK

This Chapter will guide you through the steps which are necessary to do code panel code checking using CSR Tank. A reference case will be used; the slice of a tanker that was introduced earlier in this user documentation.

Note that although this chapter deals with using PULS in the context of CSR Tank, it is fully possible to use PULS (make a CSR Tank run) for a general plated structure consisting of stiffened/unstiffened panels.

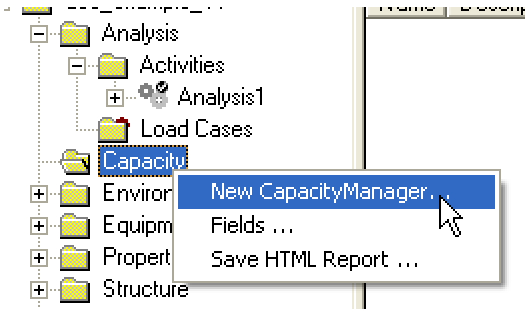
The following procedure may be used when performing code checks; each is described in detail except for the three first steps (please consult the User Manual Vol. I for guidance).

- Make a plate model. Beams may be included. A “real world” model will typically contain both beams and plates.
- Run the finite element analysis
- Define relevant load combinations if they were not part of the above analysis.
- Create a capacity manager
- Define the panels
- Create a code check run
- Assign specific settings to individual panels
- Compute the code checking forces
- Perform the code check and investigate the results graphically or from the browser.
- If necessary modify plate thickness, materials or other code checking parameters. Note that changes made to structure model requires that you re-run the FEM analysis and re-generate the panels. Local changes on some of the capacity panels are allowed, but these will be lost when re-creating the panels.
- Make a report

4.1 Create a capacity manager

The purpose of a capacity manager is to decide which analysis results to use in the code checking. It is possible to have several analysis activities in GeniE where you can have different loadcases as well as analyse subsets of your model. To be able to do code checking you need to define multiple capacity managers referring to the different analysis in question.

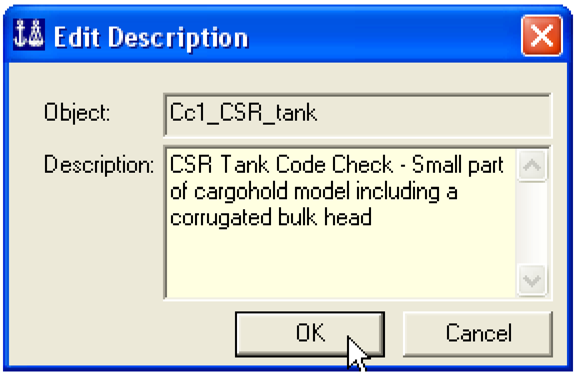
In this case we have one analysis, and one capacity manager is created from the browser.

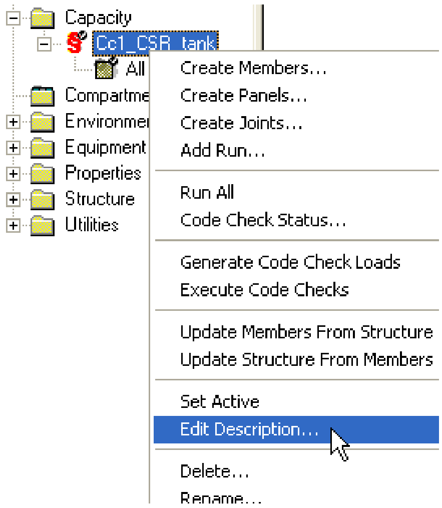


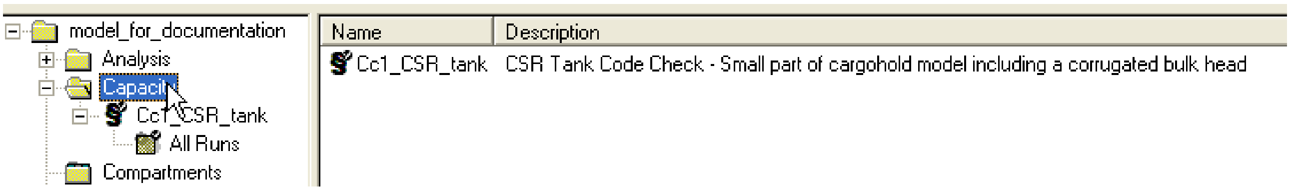
Note that when naming the capacity manager it must be given a unique name. You cannot give it the same name as a set for instance.

You may add additional description to the capacity manager. The description is also shown in the browser.

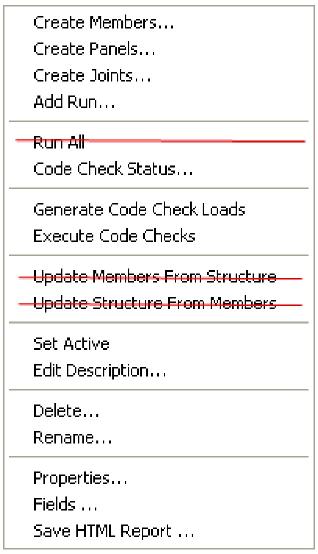
When making a report, the descriptions are also documented.







IMPORTANT LIMITATION: The menu below is common to all code checks. However the menu items *Run All*, *Update Members From Structure* and *Update Structure From Members* are intended to be used when using redesign for a member code check (beams).

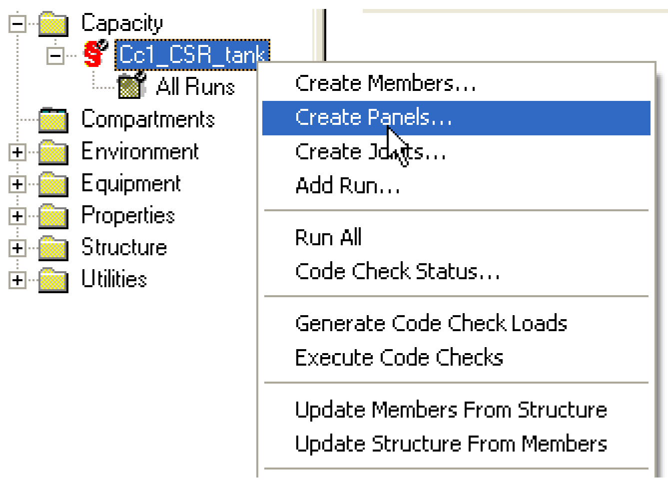


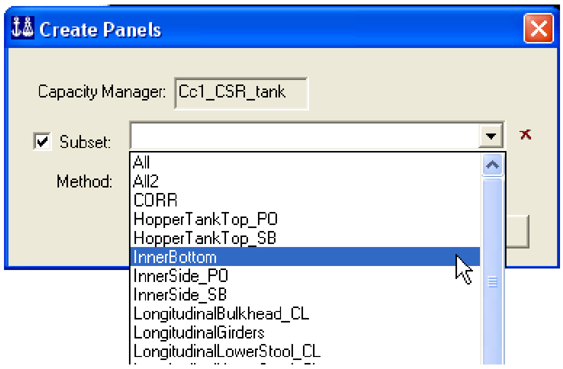
The menu items *Run All*, *Update Members From Structure* and *Update Structure From Members* should not be used for a CSR Tank run.

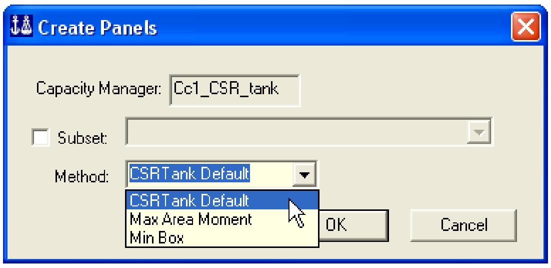
4.2 Defining panels

When modelling a concept model it is possible to make continuous plates that span several beams. This means that the concept model is different from a panel capacity model which spans between two beams only. It is therefore necessary to split up the concept model into elementary panel capacity models.

When the model is split or kept, the default buckling lengths are set since they are the same as the length or the breadth of a capacity panel.







If you are working on a large tanker it is necessary to divide the model into smaller subsets and create panels for one subset at a time. This can be done by checking the *Subset* check button and choose a subset from the subset list.

When creating panels there is 1 choice available for CSR Tank type runs:

- **CSRTankDefault: Is the algorithm to be used when preparing a CSR Tank code check. Will create planar panels.**

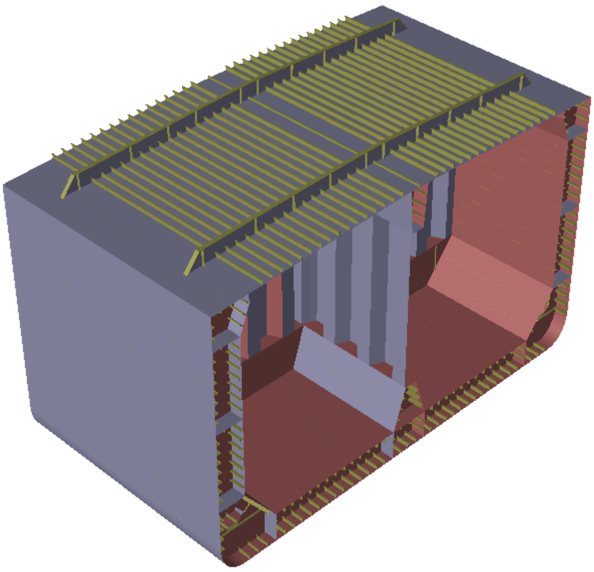
Note that the CSR_BC_OT_Default method also will create panels for curved regions. However, this method should not be used for subsequent execution with the CSR Bulk or CSR Tank code-check, since these code-checks in GeniE do not include specific computations for panels created from curved regions. If however, this method is used with CSR Bulk or CSR Tank code-check, the validity of the results must be evaluated by the user – the more panel curvature increases the more non-consitent results.

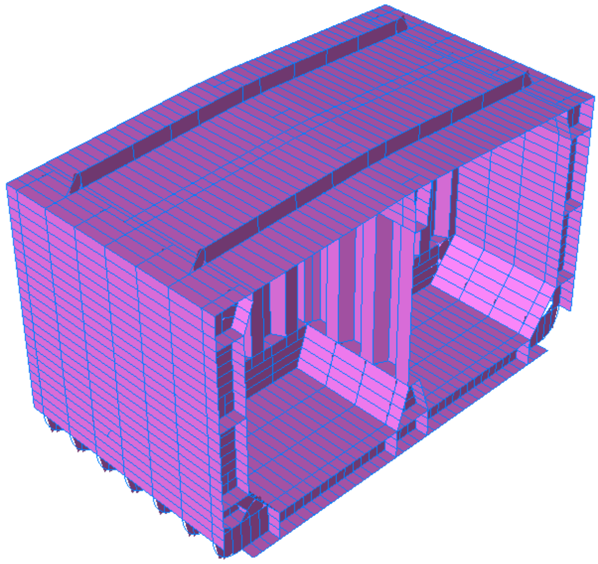
The other panel creation methods will only accept slighty curved regions, within 1 degree of curvature, and only create planar panels.

Note that CSR Tank Default will not cover panel generation for some panels made from plates with a unusual geometry. In that case GeniE will automatically pick another panel generating method.

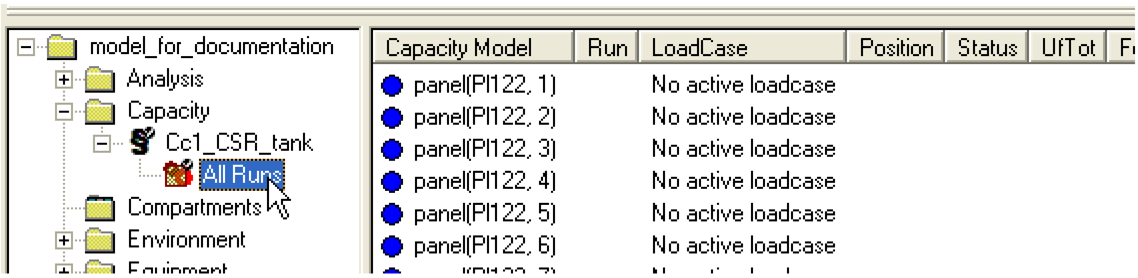
The naming convention of capacity panels refer to the plates. If Plate *Pl20* is split into two capacity panels they are denoted *panel(Pl20,1)* and *panel(Pl20,2)*.

The pictures show the concept model and the capacity panels.





The browser lists each capacity panel. If you have several capacity managers you need to specify which manager is active (select a manager, RMB and choose *Set Active*). In this case “CapMan_CSR_Bulk” is the only capacity manager and it is set to active.

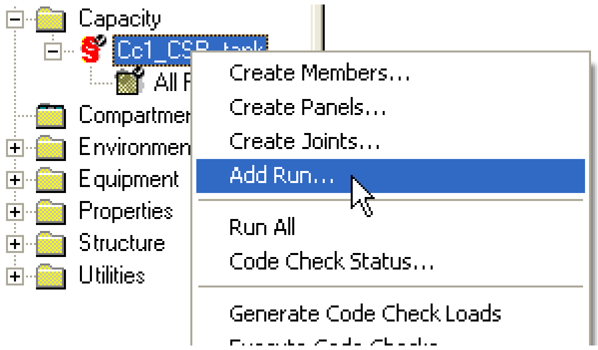


4.3 Create a CSR Tank code check run

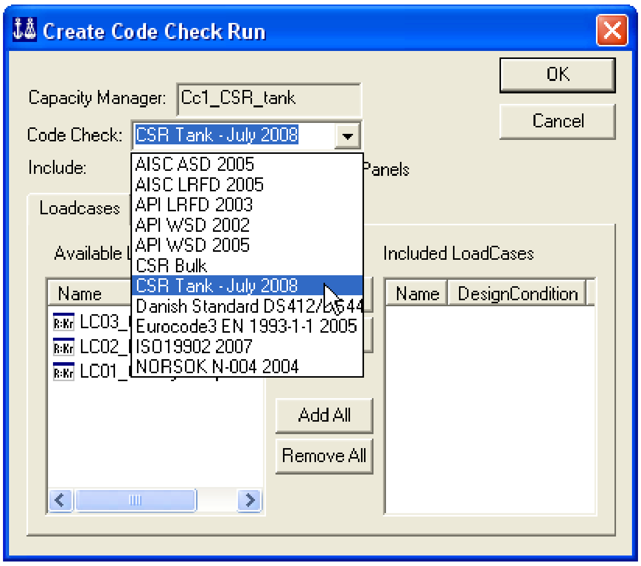
During the definition of code check runs you decide

- which code of practice to use
- which loadcases to use
- global code checking parameters (i.e. those who apply to the entire capacity model) , for example code check settings, safety factors.

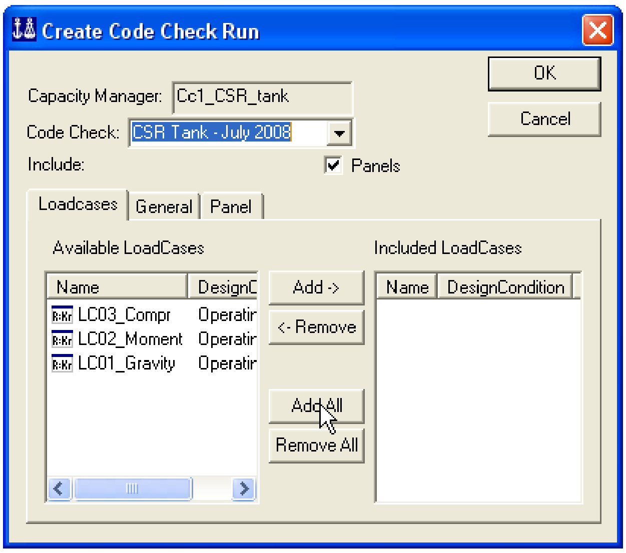
The code check run is added from the browser.

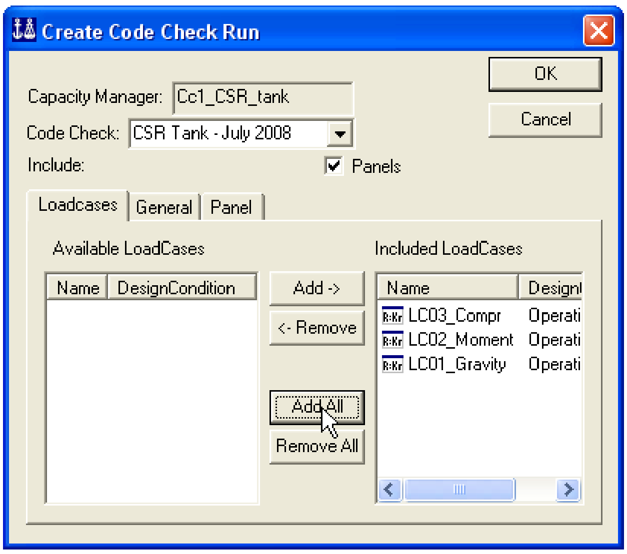


In the example to the right the code check CSR Tank has been selected.

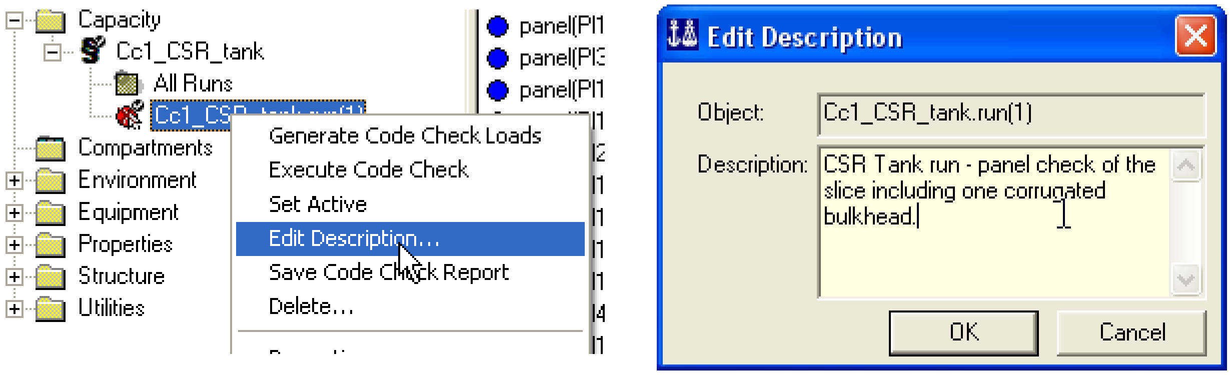


Furthermore, all the loadcases have been added to the CSR Tank code check run.

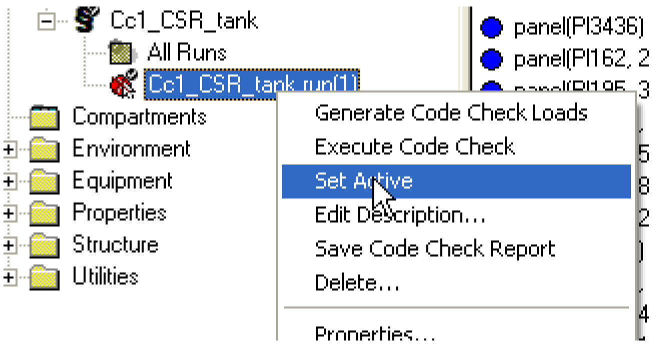




You may modify the code check run from the browser and define additional information to the run.

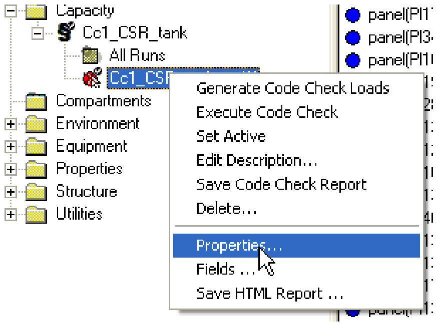


The same procedure may be used to create another code check run. You then have to specify which one is active from the browser.



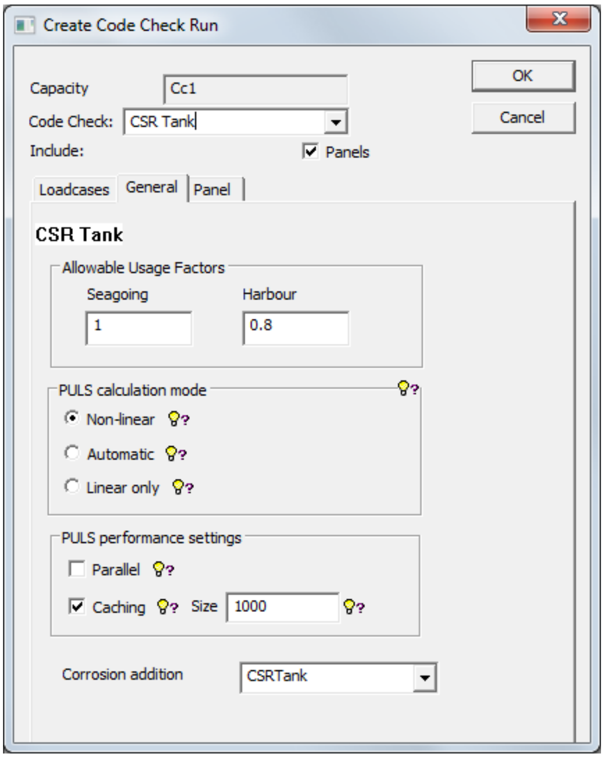
4.3.1 Define global general code checking parameters

You specify and modify the general code checking parameters when you define the code check run, or you may modify at a later stage. To modify, select *Properties* as shown on the picture to the right.



For a further description of parameters shown herein, please consult the relevant codes of practices. The general code checking parameters are global, i.e. they apply to all capacity panels.

4.3.1.1 *General parameters CSR Tank*



The general parameters for the offshore code check CSR Tank are shown below.

Allowable Usage Factors

The Allowable Usage Factors are user defined and represent the usage factors considered on operation condition (**Seagoing** and **Harbour**). The definition of these values will set the threshold values for the code check result values.

PULS Calculation Mode

The calculation mode radio buttons allow you to select the structural PULS calculation mode.

Non-linear runs a full PULS non-linear check for all capacity models. This mode is time consuming but provides the most detailed results.

Automatic the program decides whether to use linear or non-linear analysis. The decision is based on threshold values of von Mises stresses and previous linear analysis usage factor values:

By default linear analysis is carried out. The PULS non-linear calculation is used when the applied stresses exceed 80% of the von Mises stresses and/or exceed 80% of the global or local eigenvalues. In case of lateral pressure load, the limit for von Mises stresses drops to 50%. This Automatic procedure (or threshold values) remains to be approved by the class societies.

Linear only run linear calculations of PULS without detailed results just indicating the status regarding relative usage factors (OK, Failed). Note that a pure linear run may be unconservative.

PULS performance settings

PULS allow the user to set some advanced settings in order to perform time preserving calculations.

Parallel utilize multiple core processors, this makes the calculations run faster. You may experience that your computer almost stops responding during the calculation because GeniE is using all available processor cores. Check this if you don't need to do other work on your computer while the calculation runs.

Note that the amount of time used on calculation can be significantly lowered by ticking this off. On a computer with 8 cores the calculations will run almost 8 times faster.

Caching reduces memory consumption.

Size - the value indicates the total number of cases to be checked being given by the number of loadcases times number of panels. You are advised to use the default value of 1000.

Corrosion addition

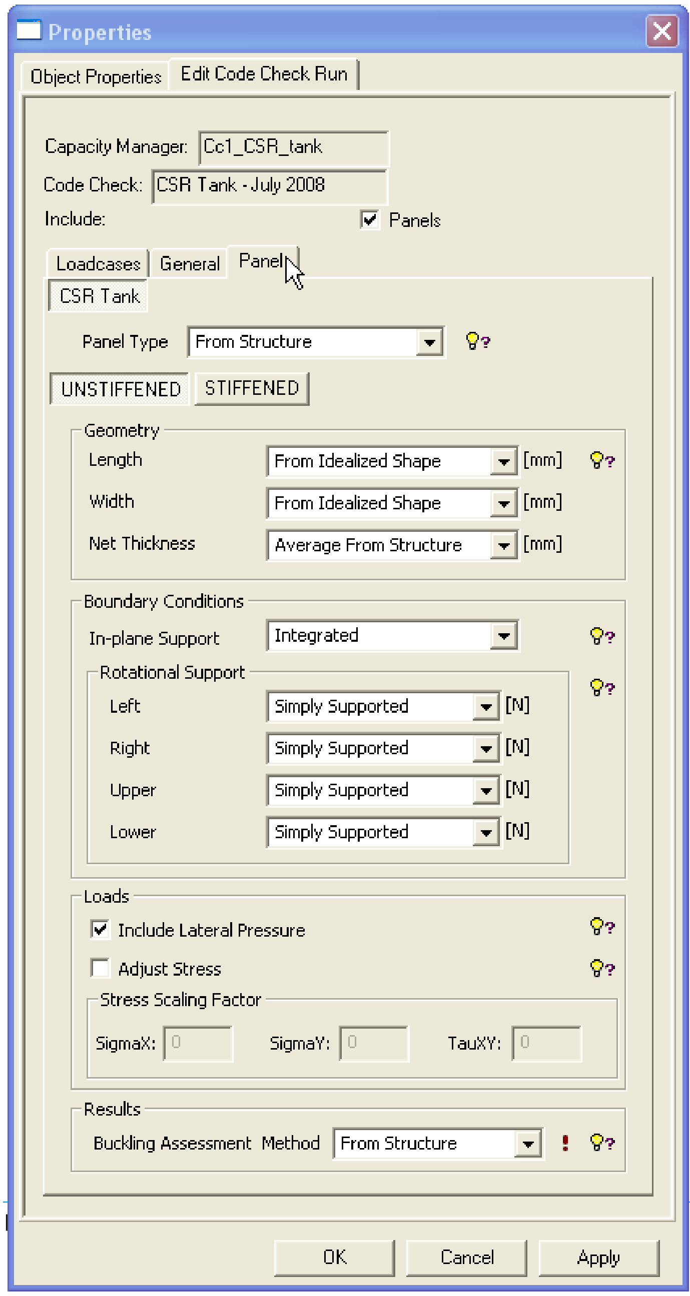
See the description for CSR BC & OT in 5.4.1.1

4.3.2 Define global panel parameters CSR Tank

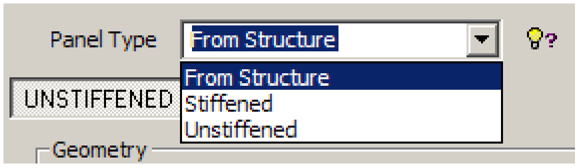
The global panel parameters (those who apply to the all capacity panels) may be changed from default values when you define the code check run or later. The default values are shown in the following for the CSR Tank code check alternative.

4.3.2.1 Panel parameters CSR Tank - unstiffened panel

The default panel options for unstiffened panels are shown below. The options for stiffened panels are presented in the next paragraph.



Panel Type defines the implemented methods in order to define panel type.



From Structure: by selecting this method all panels covered by this run are determined according to CSR Tank/App D.

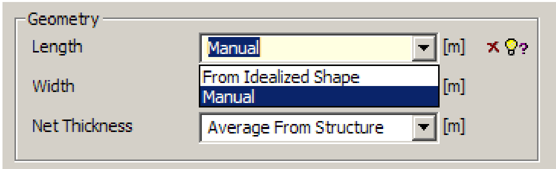
Stiffened: forces, if possible, all panels in this run to be considered as stiffened.

Unstiffened: forces all panels in this run to be considered as unstiffened.

To see/change settings for all unstiffened panels, the "Unstiffened" button needs to be pushed down.

Geometry

Length/Width



From Idealized Shape - use the values obtained by the panel simplification methods previously described in this document (in this case usually CSR Tank).

Manual – insert your own value to override the panel dimensions set by the code check. You will normally not use this.

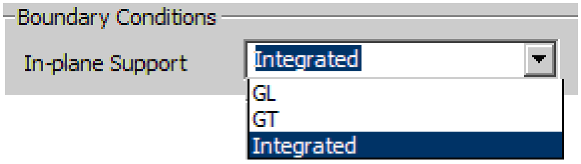
Net Thickness – the thickness of the panel when corrosion addition is subtracted.

Average from structure – Net thickness is derived from structure. If the panel consists of several concept plates with different thicknesses, an average thickness is calculated.

Manual - insert your own value to override the panel net thickness set by the code check. You will normally not use this.

Boundary Conditions

In-plane support – defines the in-plane membrane supports.

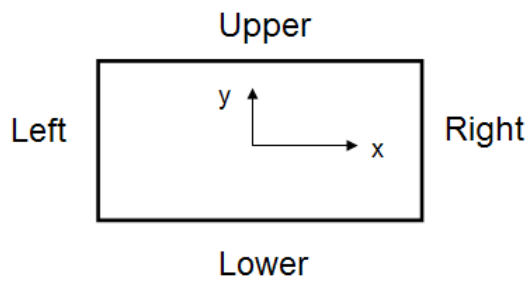


Integrated means that the panel's edges are restrained in-plane support

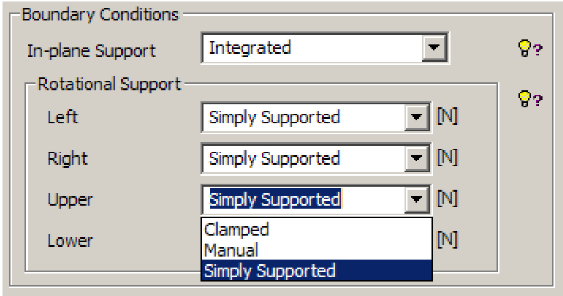
GL sets the left and right edges free.

GT sets the upper and lower edges free.

The x-axis is always running along the panel's longest edge as shown in the illustration. "Upper" and "Lower" are always the longest edges. "Left" and "Right" are always the shortest edges.



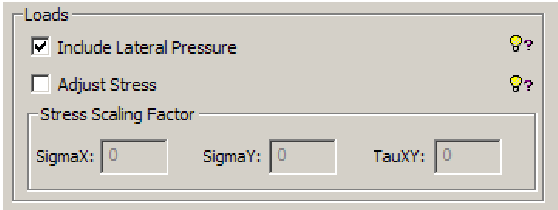
Rotational Support



Clamped/Simply Supported are predefined typical boundary conditions.
Manual – here you can enter your own value. This value should be understood as the physical properties of a rotational spring and takes the units (N*m/m). (the units might be different if you have altered the units for force and length).

Loads

It is important to understand the difference between *in-plane loads* and *out-of-plane loads*. The in-plane loads are extracted from the previous linear structural analysis and processed accordingly to match the input criteria for PULS.



Include Lateral Pressure

Out-of-plane load signifies the inclusion of lateral pressure. By default this option is selected.

Adjust Stress

It is possible to increase/decrease the stress loads. The reasons for doing this adjustment can be, for instance, to see what a change of a panel's thickness will do for the panel results.

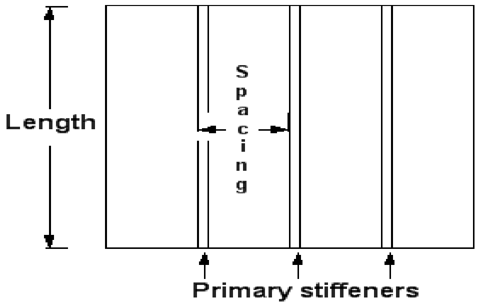
The **stress scaling factor** (*f*) is a non-dimensional parameter between 0 and 1. In case of a change of thickness (dT), from the original thickness (T), the ratio between new stress value and old stress value – stress ratio (σ_R) can be expressed on the following way:

$$\sigma_R = \frac{T}{T + f \cdot dT}$$

Buckling assessment method

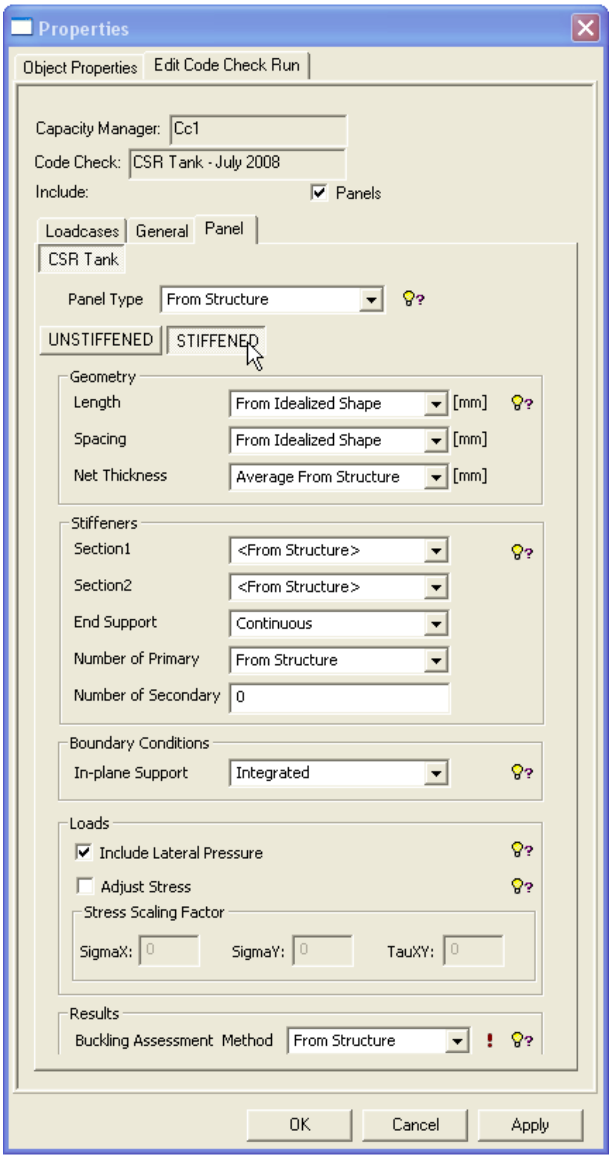
From Structure - The buckling assessment method is determined according to CSR Tank based on the Structure type of the structure.
Method 1 – Ultimate capacity
Method 2 – Buckling strength.

4.3.2.2 Panel parameters CSR Tank - stiffened panel

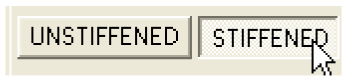


This illustration shows a stiffened panel and is useful for understanding terms explained later in this paragraph.

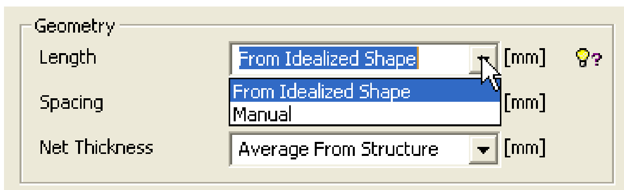
The default panel options for stiffened panels are shown below.



Panel Type is explained in the previous paragraph.



To see/change settings for all stiffened panels, the "Stiffened" button needs to be pushed down.



Geometry

Length

From Idealized Shape - use the values obtained by the panel simplification methods previously described in this document (in this case usually CSR Tank).

Manual – insert your own value to override the panel dimensions set by the code check. You will normally not use this.

Spacing - the spacing between the stiffeners

From Idealized Shape - use the values obtained by the panel simplification methods previously described in this document (in this case usually CSR Tank).

Manual – insert your own value to override the spacing between stiffeners set by the code check.

Net Thickness – the thickness of the panel when corrosion addition is subtracted.

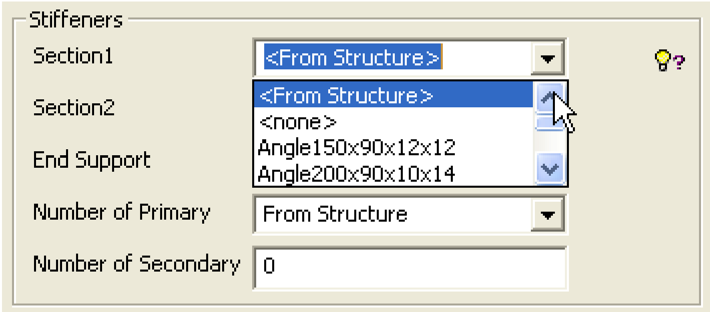
Average from structure – Net thickness is derived from structure. If the panel consists of several concept plates with different thicknesses, an average thickness is calculated.

Manual - insert your own value to override the panel net thickness set by the code check. You will normally not use this.

Stiffeners

Section1 – is the cross section of the stiffener on one of the sides of the panel.

Section2 – is the cross section of the stiffener on the other side of the panel.



<From Structure> the section is derived from the section in the structure.

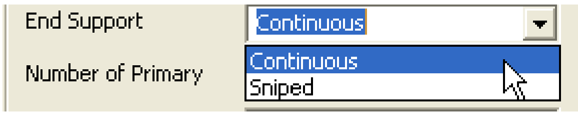
<none> indicates that no section was found.

You can manually decide upon a section from the list. The list includes all available sections in your particular Genie model.

End Support

Continuous (default)

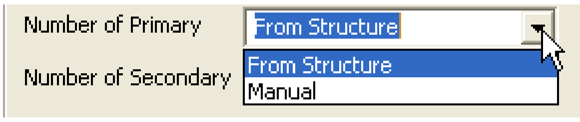
Sniped.



Number of primary

From Structure – Number of primary stiffeners is derived from structure.

Manual - insert your own value to override the number of primary stiffeners derived from the structure.



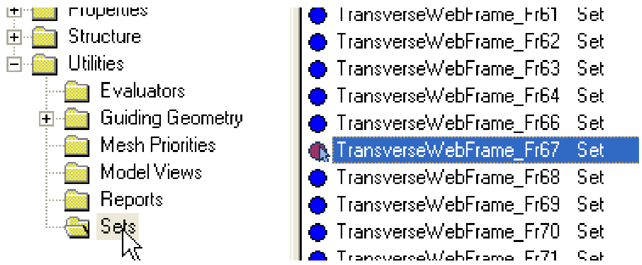
Number of Secondary – indicates the number of secondary stiffeners in the stiffened panel.

The rest of the terms in this dialog are described in the unstiffened panel description in the previous paragraph.

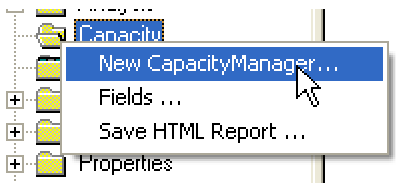
4.4 Make a capacity manager and a run for one specific set and performing the code check.

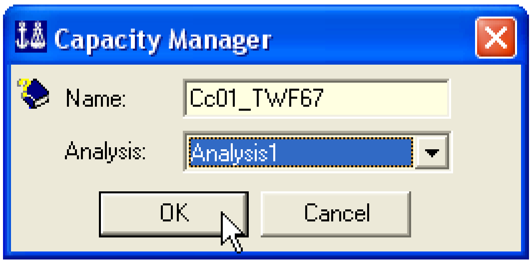
When working with CSR Tank on large models it is generally impractical to work with the entire model in one capacity manager. To keep an overview and for calculations not to be too time consuming, you should make a separate capacity manager with one run for each set of interest.

By looking at the list of Sets in the browser we find a set of interest, for instance *TransverseWebFrame_Fr67*.

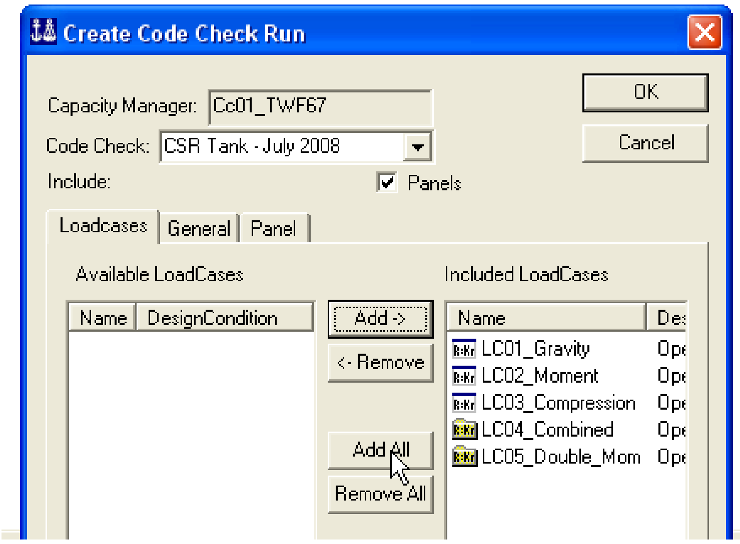


You make a new capacitymanager and give it a name that easily identifies which set is going to be handled in this capacitymanager.

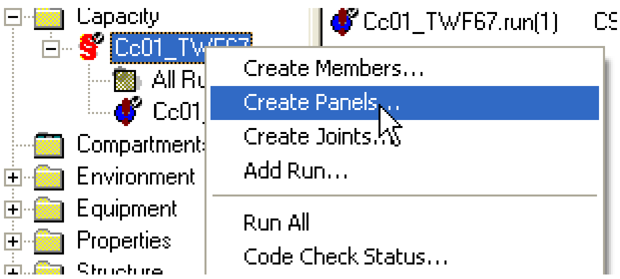


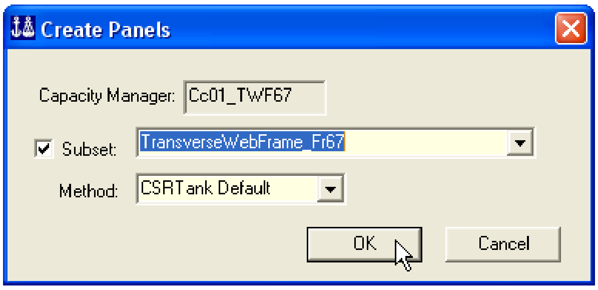


You add a run, choose CSR Tank as code check and include the loadcases of interest. In the illustration all loadcases have been selected.

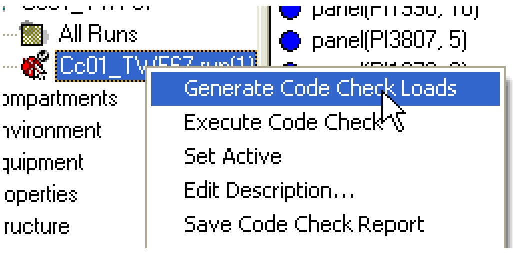


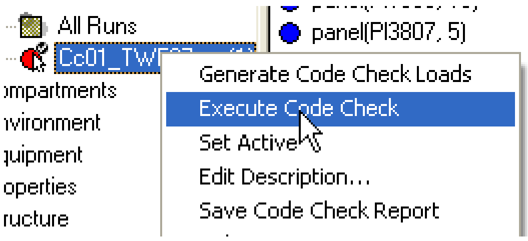
Select *Create Panels* from the **RMB** menu and make sure the Subset checkbutton is ticked off and that your set of interest is selected. Also make sure to select CSRTank Default in the Method drop down menu.



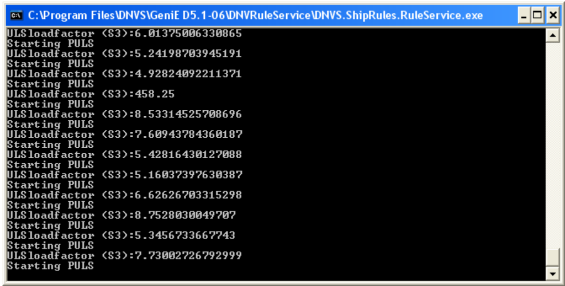


RMB click on your run, select *Generate Code Check Loads* and then select *Execute Code Check.*





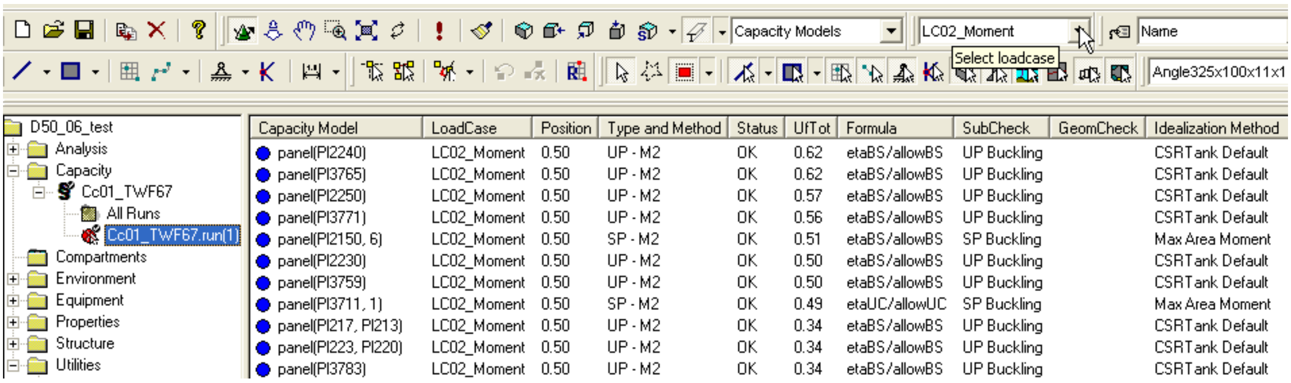
When you have selected *Execute Code Check* (for a CSR Tank run) PULS starts running in a separate console window, similar to the one to the right. Observe that the calculation is running as long as new numbers keep showing up from the bottom of the console window.



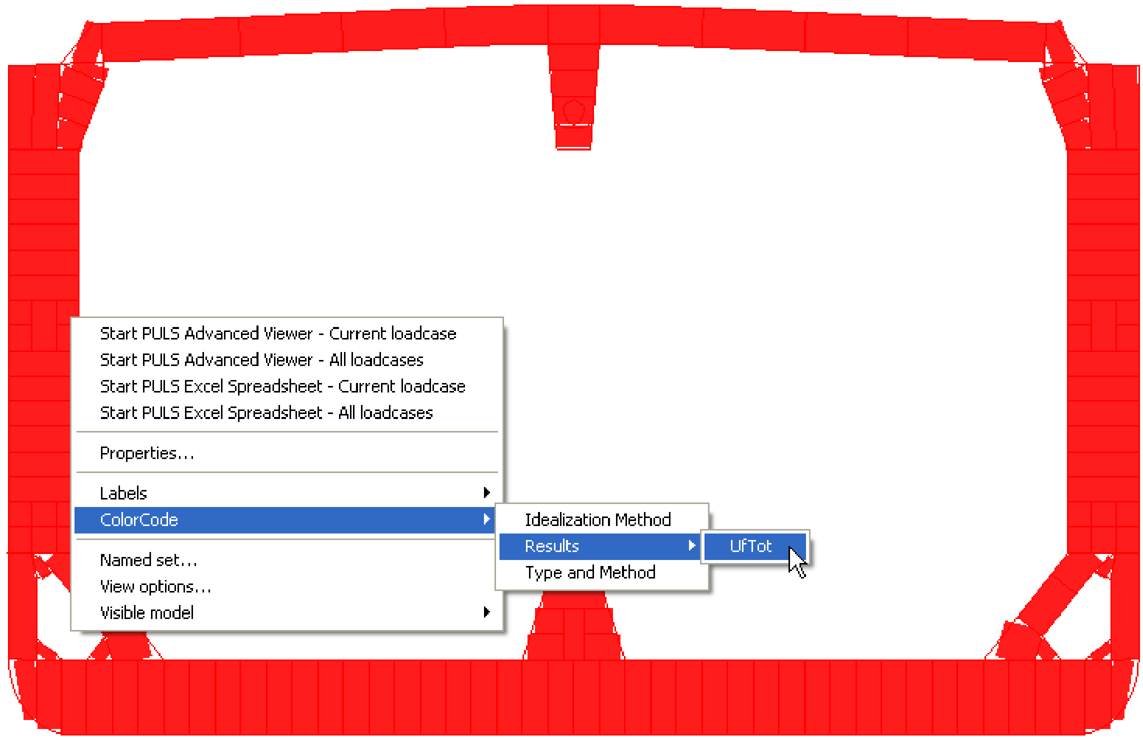
Running PULS is often quite time consuming. If your computer has several processor cores, you could speed up the calculation by ticking off in the checkbox "Parallel" under PULS Performance Settings as shown in paragraph 4.3.1.1.

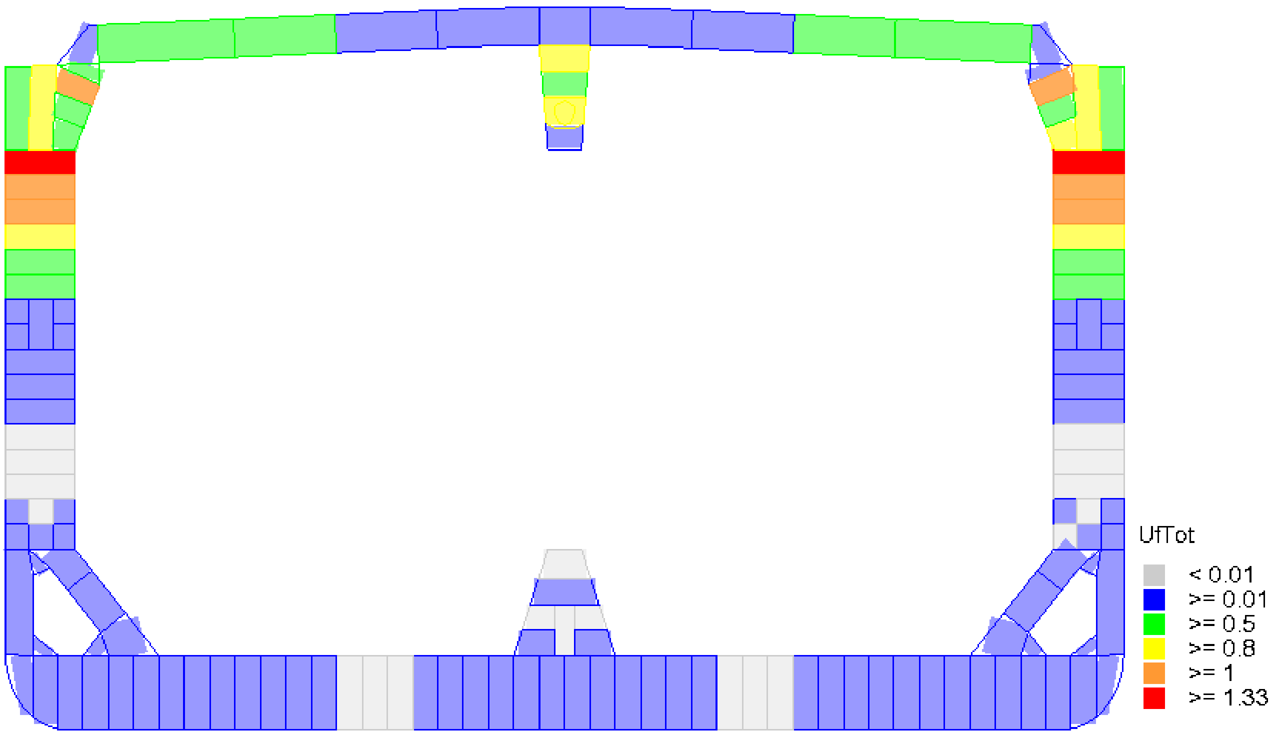
The console window closes itself and disappears when the calculation is completed.

The results are now available in the browser:



The results can also be color coded on the capacity model in the graphical window. Select the panels of interest, and click **RMB** in the graphical view to color code UfTot

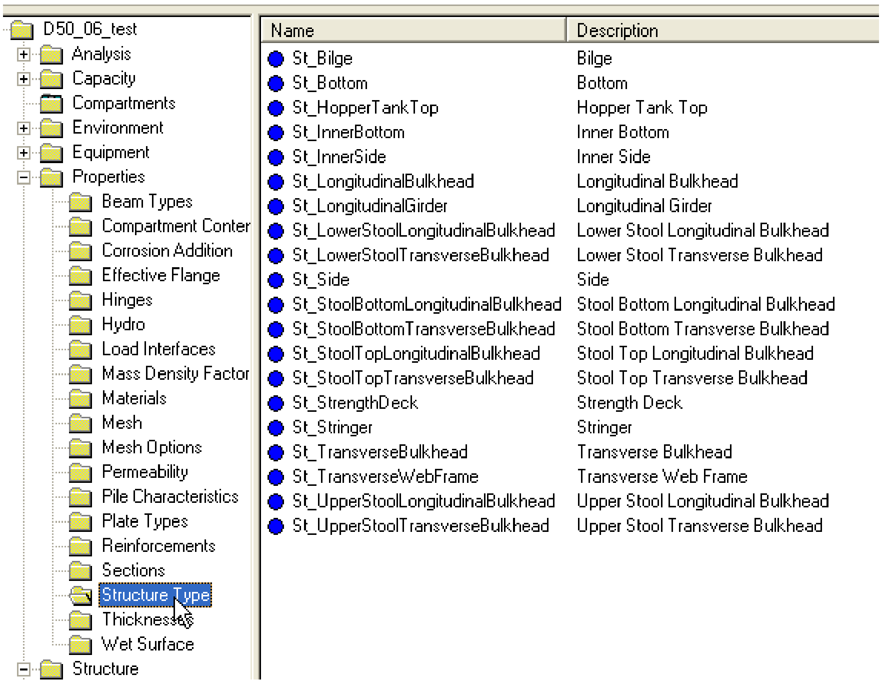




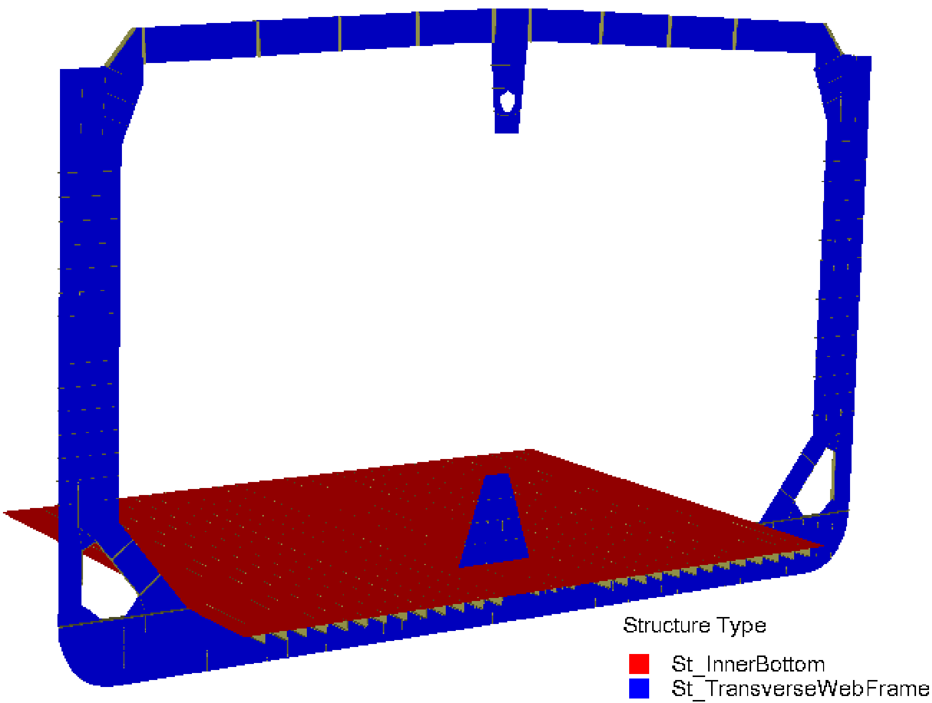
You should repeat the steps in this paragraph for all sets that you want to check.

4.5 Structure types and methods

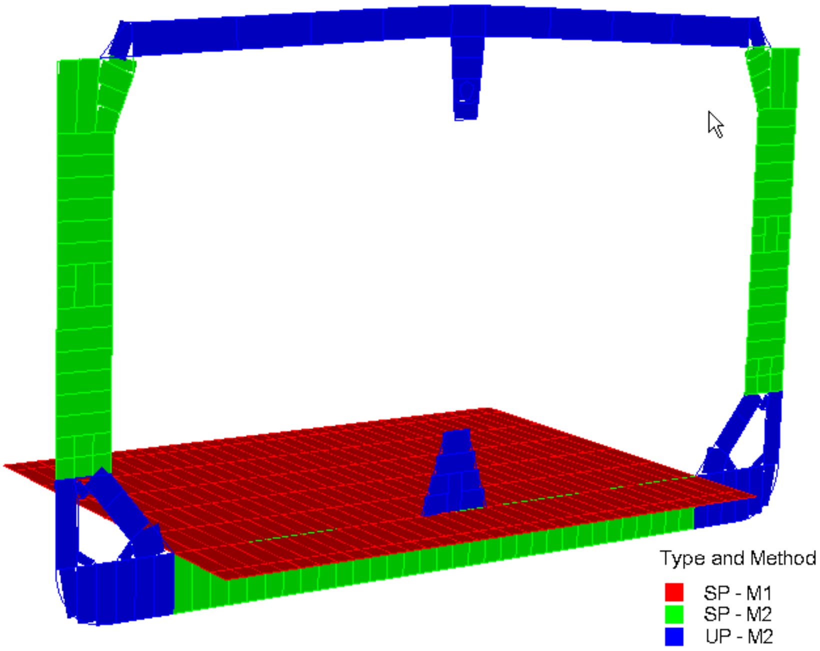
When importing the xml file created by NH, the model will have different structure types corresponding to the different parts of the model. All structure types that are used for your model can be viewed in the browser by clicking *Structure Type* under *Properties*.



You can color code the structure types to check that the correct structure types are applied to the correct parts of the model. The CSR Tank buckling check will use the structure type information to decide whether the different parts of the model should be represented as stiffened or unstiffened panels.

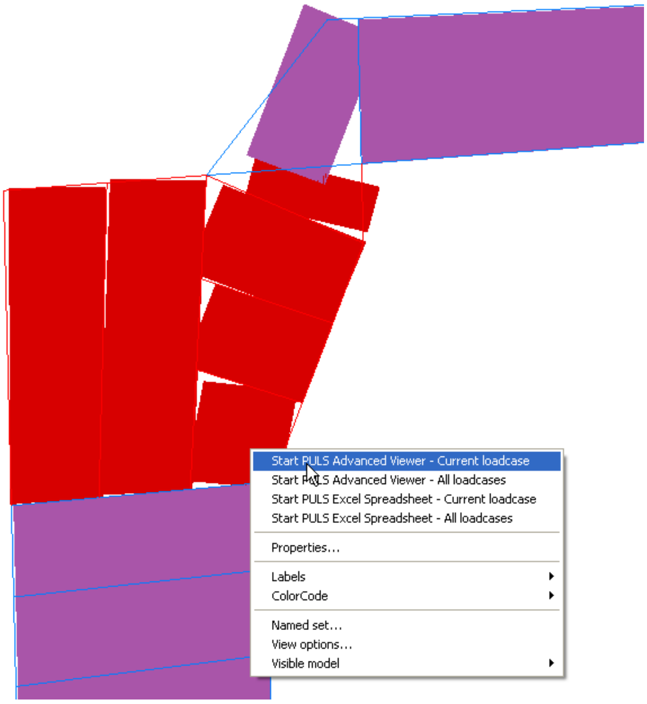


Type and method are decided based on structural type and also from the panels' location as stated in App D in the CSR Tank rules.



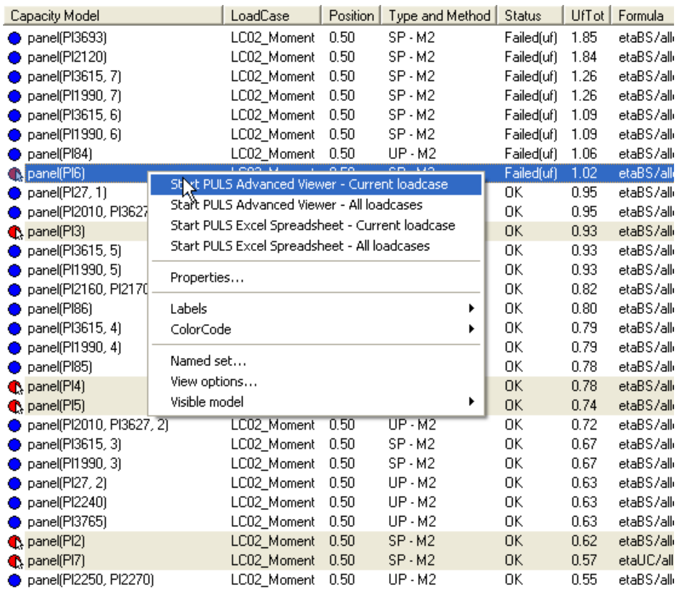
4.6 Starting PULS Advanced Viewer and PULS Spreadsheet from GeniE

You can start PULS Advanced Viewer and PULS Excel Spreadsheet from inside GeniE. This can be done either from the graphical window or from the browser. Note that these components are only available as part of an installation of Nauticus Hull.



In the graphical window you select one ore several panels, click **RMB** and make your choice. You can choose to open the panel(s) only for the current loadcase or for all loadcases.

From the browser you can select one or several panels, click **RMB,** and the same menu shows up.



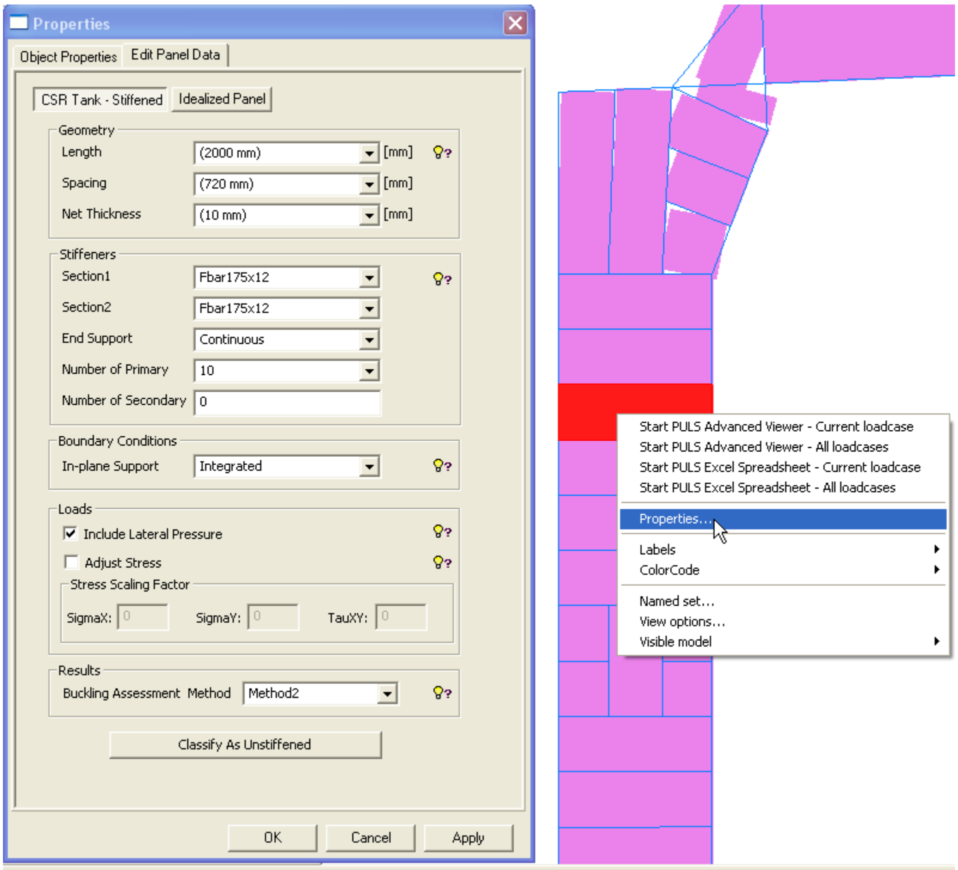
For more information on usage of the PULS Advanced Viewer and the PULS Spreadsheet consult the Nauticus Hull user manuals.

4.7 Changing properties for a panel

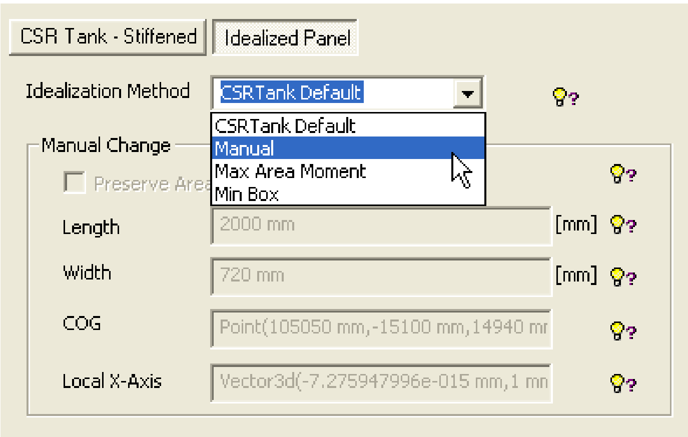
By selecting one or several panels, clicking **RMB** and selecting *Properties*, you get to see the properties of the panel(s) you have selected.

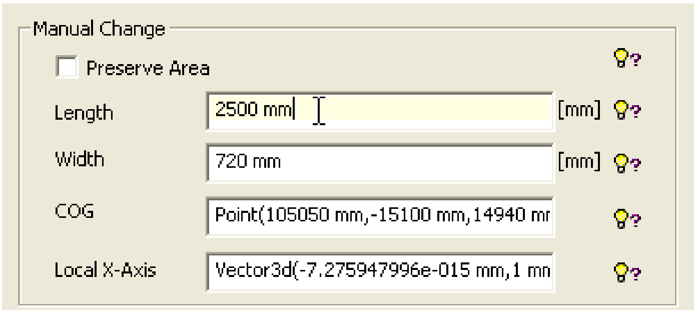
In the example illustration, you can see the properties for one panel.

The values for Length, Spacing and Net Thickness are shown in brackets. This means that these values are based on the idealized panel.



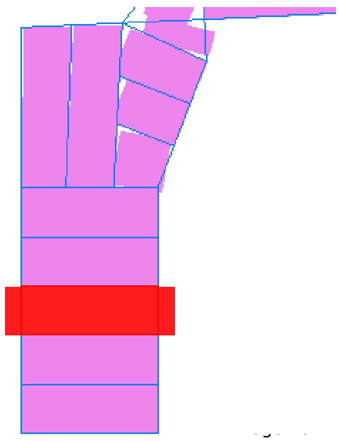
By clicking on “Idealized Panel” you get to change some of the values for the Idealized Panel. The changes you do will only affect the selected panel(s).

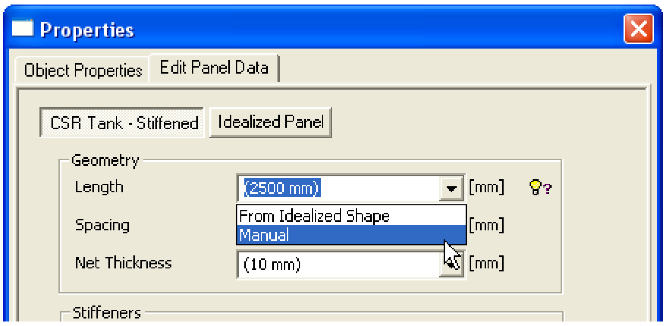




You are able to select idealization method from the available panel idealization methods, CSR Tank, Max Area Moment and Min Box. In addition you are able to manually set the properties of the panel(s). In this example we are manually changing the length of one panel.

Changes will be visible in the graphical display.





It is also possible to change the properties for the selected panel(s) directly without changing the idealized panel settings.

Geometry

Length/Spacing

From Idealized Shape – Uses the dimensions taken from the properties of the idealized panel. The numbers are shown with brackets around them to indicate they are the same as for the idealized panel.

Manual – You can set your own value manually.

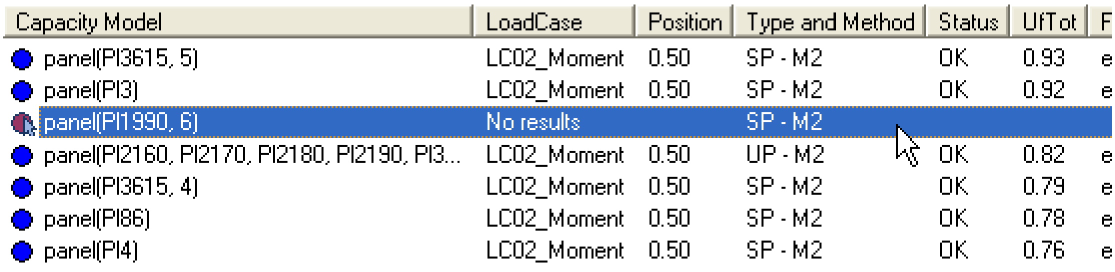
Net Thickness

Average from Structure – The net thickness is calculated from the thickness(es) of the conceptual plates from which the panel(s) are derived.

Manual – You can set your own net thickness value manually.

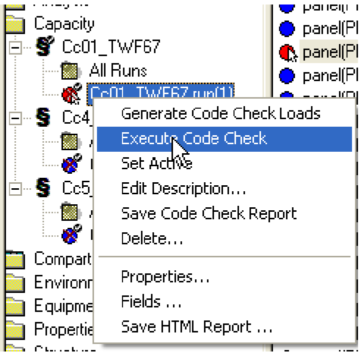
The rest of the properties can be changed in a similar way.

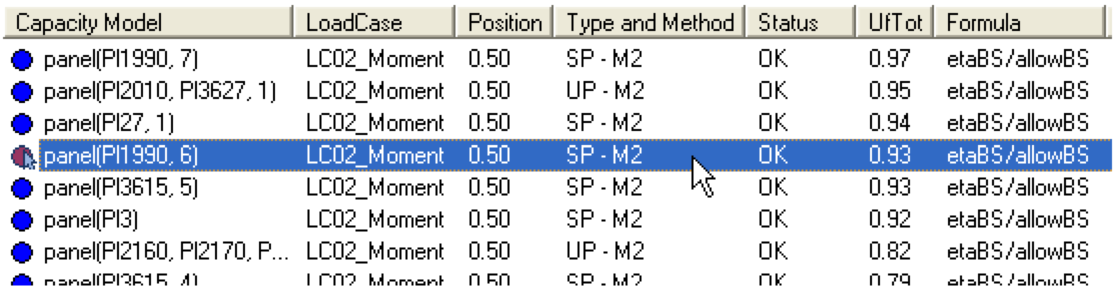
Note that the panels which properties have been change will show up with "No results" in the browser.



Capacity Model	LoadCase	Position	Type and Method	Status	UfTot	F
panel(Pl3615, 5)	LC02_Moment	0.50	SP - M2	OK	0.93	e
panel(Pl3)	LC02_Moment	0.50	SP - M2	OK	0.92	e
panel(Pl1990, 6)	No results		SP - M2			
panel(Pl2160, Pl2170, Pl2180, Pl2190, Pl3...	LC02_Moment	0.50	UP - M2	OK	0.82	e
panel(Pl3615, 4)	LC02_Moment	0.50	SP - M2	OK	0.79	e
panel(Pl86)	LC02_Moment	0.50	SP - M2	OK	0.78	e
panel(Pl4)	LC02_Moment	0.50	SP - M2	OK	0.76	e

To see any results for the panels you need to select *Execute Code Check* for the appropriate run.

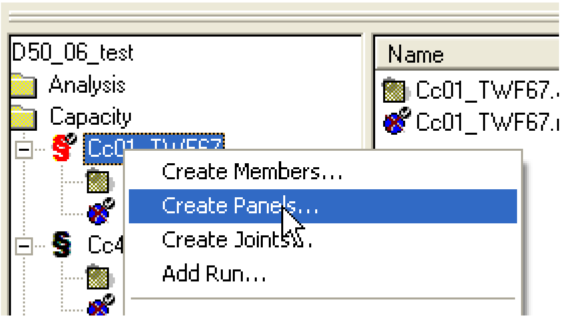




Capacity Model	LoadCase	Position	Type and Method	Status	UfTot	Formula
panel(Pl1990, 7)	LC02_Moment	0.50	SP - M2	OK	0.97	etaBS/allowBS
panel(Pl2010, Pl3627, 1)	LC02_Moment	0.50	UP - M2	OK	0.95	etaBS/allowBS
panel(Pl27, 1)	LC02_Moment	0.50	SP - M2	OK	0.94	etaBS/allowBS
panel(Pl1990, 6)	LC02_Moment	0.50	SP - M2	OK	0.93	etaBS/allowBS
panel(Pl3615, 5)	LC02_Moment	0.50	SP - M2	OK	0.93	etaBS/allowBS
panel(Pl3)	LC02_Moment	0.50	SP - M2	OK	0.92	etaBS/allowBS
panel(Pl2160, Pl2170, P...	LC02_Moment	0.50	UP - M2	OK	0.82	etaBS/allowBS
panel(Pl3615, 4)	LC02_Moment	0.50	SP - M2	OK	0.79	etaBS/allowBS

Important

The purpose of making changes to one or several panels as described in this paragraph is to test how changing the different properties will affect the usage factor etc for the panel(s). To permanently change the properties for the panels, you need to make changes to the concept model.



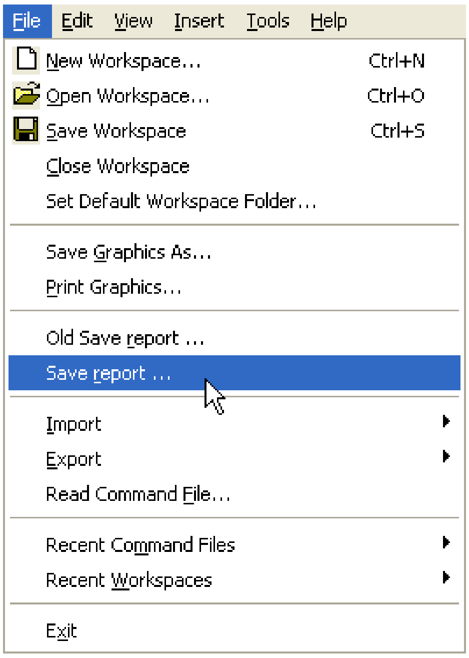
Note that if you select *Create Panels* for your run, all changes you have done to the panels will disappear.

4.8 Make a report CSR Tank

A customised report may be generated from the ***File|Save report***. Please notice that the report functionality in versions prior to GeniE version v3.4-27 is still available from the command ***File|Old Save report***.

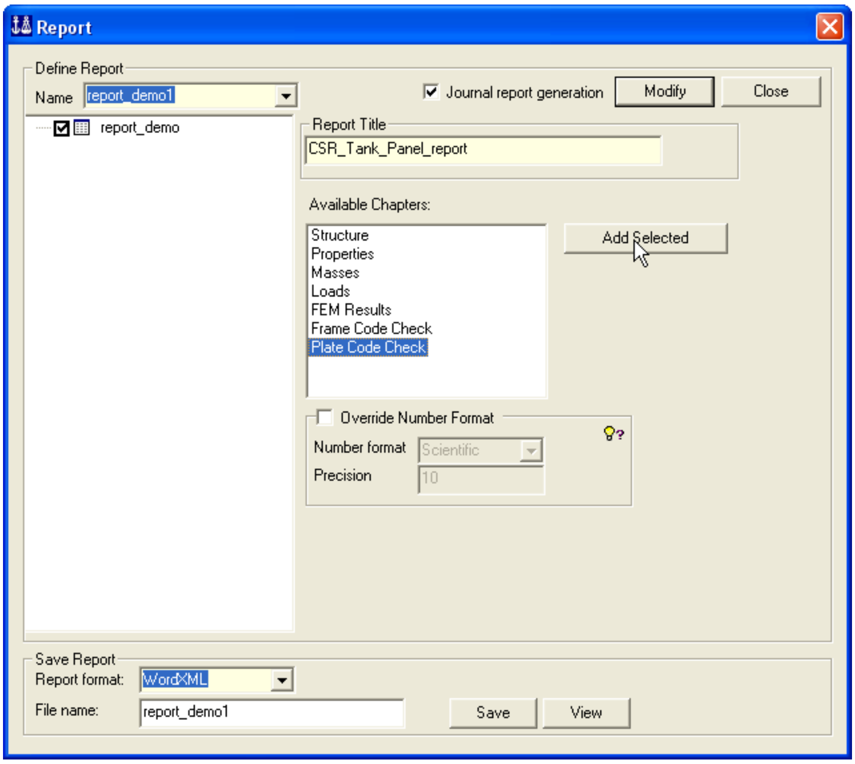
The **File|Save report** allows you to specify the content of your report as well as to decide the levels of detail in of your report.

In the following is given an example on how to make a code checking report.



To make a report of the plate code check, you select *Plate Code Check* under *Available Chapters* and click *Add Selected*.

The report name, title and file name can be changed. By default the file name is the same as the report name.



The available tables for plate code check are *Panel Options Full*, *Panel Result Brief* and *Panel Result Full*.

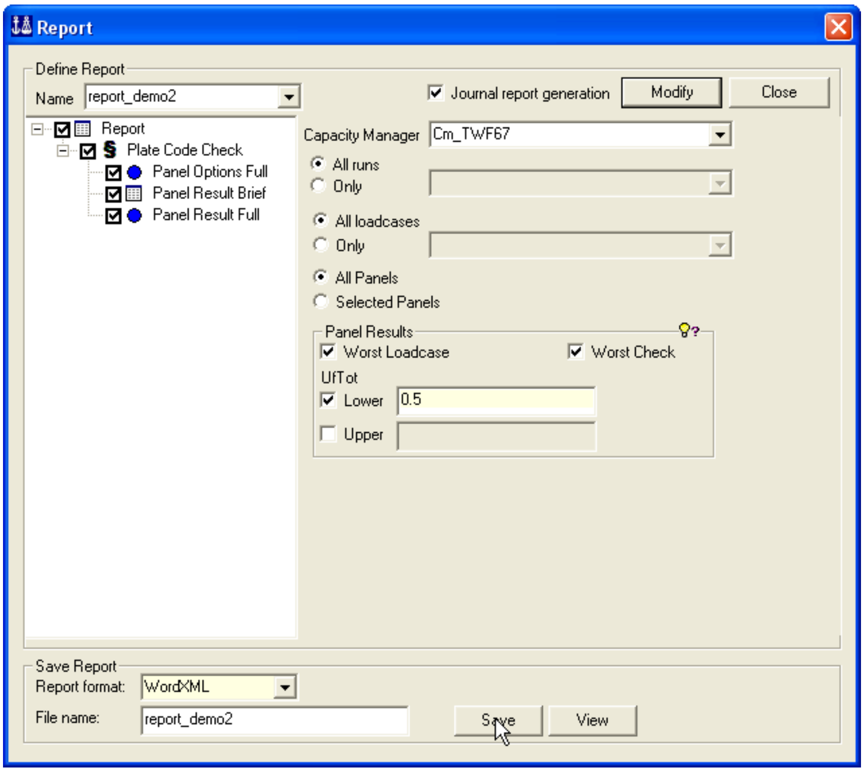
We have checked the *Lower* checkbox for Uftot, and typed in the value 0.5.

This means that the report will not list any panels having a Uftot at 0.5 or lower.

We have chosen the report format to be *WordXML*. Other available formats are: *HTML*, *ExcelXML* and *Text*.

After having clicked *Save*, you can have a look at the report by clicking *View*.

Below are examples of what our selected tables might look like in the report.



Panel Options Full, Unstiffened Panels:

1.1.1.1 Cm_TWF67.run(1) : Unstiffened Panel Options CSR Tank

Cm_TWF67.run(1) : Unstiffened Panel Options CSR Tank

- Sorted by Panel (Ascending)

- Run :Cm_TWF67.run(1)
- Only panels with options modified from the Default object(s) will be listed

Panel	Code	Panel Length [m]	Panel Width [m]	Net Thickness [m]	In-plane Support	Rotation Support Left [N]	Rotation Support Right [N]	Rotation Support Upper [N]	Rotation Support Lower [N]
	Lateral Pressure Included	**Adjust Stress**	**SigmaX ScalingFactor**	**SigmaY ScalingFactor**	**TauXY ScalingFactor**	**Governing Method**	**Idealization Method**		
Default	CSR Tank Unstiffened	From Idealized Shape	From Idealized Shape	Average From Structure	Integrated	Simply Supported	Simply Supported	Simply Supported	Simply Supported
	true	false	0	0	0	From Structure			
Pl217, Pl213	CSR Tank Unstiffened	(2.945828415 m)	(1.086238154 m)	(0.01324450733 m)	Integrated	Simply Supported	Simply Supported	Simply Supported	Simply Supported
	true	false	0	0	0	Method2	CSRTank Default		
Pl218, Pl219	CSR Tank Unstiffened	(1.46 m)	(1.1 m)	(0.0155 m)	Integrated	Simply Supported	Simply Supported	Simply Supported	Simply Supported
	true	false	0	0	0	Method2	CSRTank Default		

Panel Options Full, Stiffened Panels:

1.1.1.2 Cm_TWF67.run(1) : Stiffened Panel Options CSR Tank

Cm_TWF67.run(1) : Stiffened Panel Options CSR Tank

- Sorted by Panel (Ascending)

- Run : Cm_TWF67.run(1)
- Only panels with options modified from the Default object(s) will be listed

Panel	Code	Panel Length [m]	Spacing [m]	Net Thickness [m]	In-plane Support	Section 1	Section 2	End Support	Number of Primary
	Number of Secondary	**Lateral Pressure Included**	**Adjust Stress**	**SigmaX ScalingFactor**	**SigmaY ScalingFactor**	**TauXY ScalingFactor**	**Governing Method**	**Idealization Method**	
Default	CSR Tank Stiffened	From Idealized Shape	From Idealized Shape	Average From Structure	Integrated	From Structure	From Structure	Continuous	From structure
	0	true	false	0	0	0	From Structure		
Pl2	CSR Tank Stiffened	(2.376966001 m)	(0.6945623113 m)	(0.009 m)	Integrated	<none>	Fbar175x12	Continuous	5
	0	true	false	0	0	0	Method2	CSRTank Default	
Pl3	CSR Tank Stiffened	(2.412129463 m)	(0.7184966758 m)	(0.009 m)	Integrated	Fbar175x12	Fbar175x12	Continuous	5

Panel Result Brief:

1.1.2 Cm_TWF67.run(1) : Panel Result Brief

Cm_TWF67.run(1) : Panel Result Brief

- Sorted by UfTot (Descending)
- Then sorted by LoadCase (Ascending)

- Run : Cm_TWF67.run(1)
- Worst LoadCase per Panel
- All SubChecks per Panel
- Worst of Whole Panel and SubPanel Checks per Panel

Panel	LoadCase	Position	Status	UfTot	Formula	GeomCheck	SubCheck	Run
Pl3693	LC05_Double_Mom	0.50	Failed(uf)	3.69	etaBS/allowBS	Geom OK	SP Buckling	Cm_TWF67.run(1)
Pl2120	LC05_Double_Mom	0.50	Failed(uf)	3.67	etaBS/allowBS	Geom OK	SP Buckling	Cm_TWF67.run(1)
Pl3615, 7	LC05_Double_Mom	0.50	Failed(uf)	2.52	etaBS/allowBS	Geom OK	SP Buckling	Cm_TWF67.run(1)
Pl3615, 6	LC05_Double_Mom	0.50	Failed(uf)	2.18	etaBS/allowBS	Geom OK	SP Buckling	Cm_TWF67.run(1)
Pl84	LC05_Double_Mom	0.50	Failed(uf)	2.05	etaBS/allowBS	Geom OK	SP Buckling	Cm_TWF67.run(1)
Pl6	LC05_Double_Mom	0.50	Failed(uf)	2.00	etaBS/allowBS	Geom OK	SP Buckling	Cm_TWF67.run(1)
Pl1990, 7	LC05_Double_Mom	0.50	Failed(uf)	1.94	etaBS/allowBS	Geom OK	SP Buckling	Cm_TWF67.run(1)

Panel Result Full, Unstiffened Panels

1.1.3 Cm_TWF67.run(1) : Panel Result Full

1.1.3.1 Cm_TWF67.run(1) : Unstiffened Panel Buckling Result CSR Tank

Cm_TWF67.run(1) : Unstiffened Panel Buckling Result CSR Tank

- Sorted by Panel (Ascending)
- Then sorted by LoadCase (Ascending)

- Run : Cm_TWF67.run(1)
- Worst LoadCase per Panel
- Selected SubCheck per Panel
- Worst of Whole Panel and SubPanel Checks per Panel

Panel	LoadCase	Position	Status	UfTot	Formula	GeomCheck	SubCheck	Run
	Panel Length [m]	Panel Width [m]	Panel Thickness [m]	EModulus [Pa]	Poisson's Ratio	Yield Stress Plate [Pa]	Sigma11 [Pa]	Sigma12 [Pa]
	Sigma21 [Pa]	Sigma22 [Pa]	Tau12 [Pa]	Pressure [Pa]	In-plane Support	SupportLeft [N]	Support Right [N]	SupportUpper [N]
	SupportLower [N]	Sigma11UC [Pa]	Sigma12UC [Pa]	Sigma21UC [Pa]	Sigma22UC [Pa]	Tau12UC [Pa]	EtaUC	AllowUC
	StatusUC	Sigma11BS [Pa]	Sigma12BS [Pa]	Sigma21BS [Pa]	Sigma22BS [Pa]	Tau12BS [Pa]	Eta BS	AllowBS
	StatusBS	Sigma11LEB [Pa]	Sigma12LEB [Pa]	Sigma21LEB [Pa]	Sigma22LEB [Pa]	Tau12LEB [Pa]	Plate Slend Req	Plate Aspect Ratio
	PULS Error Code							
Pl217, Pl213	LC05_Double_Mom	0.50	OK	0.68	etaBS/allowBS	Geom OK	UP Buckling	Cm_TWF67.run(1)
	2.94583	1.08624	0.0132445	2.06e+011	0.3	2.35e+008	2.94962e+007	2.94962e+007
	3.84231e+006	3.84231e+006	8.2624e+007	0	Int	SS	SS	SS
	SS	4.35406e+007	4.35406e+007	5.6718e+006	5.6718e+006	1.21965e+008	0.677	1.000
	OK	4.31961e+007	4.31961e+007	5.62693e+006	5.62693e+006	1.21e+008	0.683	1.000
	OK	4.31961e+007	4.31961e+007	5.62693e+006	5.62693e+006	1.21e+008	Passed	Passed
	0							
Pl218, Pl219	LC05_Double_Mom	0.50	OK	0.60	etaBS/allowBS	Geom OK	UP Buckling	Cm_TWF67.run(1)
	1.46	1.1	0.0155	2.06e+011	0.3	3.15e+008	3.06783e+007	3.06783e+007
	3.73038e+007	3.73038e+007	-156761	0	Int	SS	SS	SS
	SS	1.03695e+008	1.03695e+008	1.2609e+008	1.2609e+008	-529866	0.296	1.000
	OK	5.09351e+007	5.09351e+007	6.19356e+007	6.19356e+007	-260271	0.602	1.000
	OK	5.09351e+007	5.09351e+007	6.19356e+007	6.19356e+007	-260271	Passed	Passed
	0							

Panel Result Full, Stiffened Panels

1.1.3.2 Cm_TWF67.run(1) : Stiffened Panel Buckling Result CSR Tank

Cm_TWF67.run(1) : Stiffened Panel Buckling Result CSR Tank

- Sorted by Panel (Ascending)
- Then sorted by LoadCase (Ascending)

- Run : Cm_TWF67.run(1)
- Worst LoadCase per Panel
- Selected SubCheck per Panel
- Worst of Whole Panel and SubPanel Checks per Panel

Panel	LoadCase	Position	Status	UfTot	Formula	GeomCheck	SubCheck	Run
	Panel Length [m]	Stiffener Spacing [m]	Plate Thickness [m]	No Primary Stiffeners	Stiffener Type	Stiffener Boundary	Stiffener Height [m]	Web Thickness [m]
	Flange Width [m]	Flange Thickness [m]	Tilt Angle [deg]	No Secondary Stiffeners	EModulus [Pa]	Poissons Ratio	Yield Stress Plate [Pa]	Yield Stress Stiffener [Pa]
	Sigma1 [Pa]	Sigma21 [Pa]	Sigma22 [Pa]	Tau12 [Pa]	Pressure [Pa]	In-plane Support	Sigma1UC [Pa]	Sigma21UC [Pa]
	Sigma22UC [Pa]	Tau12UC [Pa]	EtaUC	AllowUC	StatusUC	Sigma1BS [Pa]	Sigma21BS [Pa]	Sigma22BS [Pa]
	Tau12BS [Pa]	EtaBS	AllowBS	StatusBS	Critical Failure Mode	Sigma1GEB [Pa]	Sigma21GEB [Pa]	Sigma22GEB [Pa]
	Tau12GEB [Pa]	Sigma1LEB [Pa]	Sigma21LEB [Pa]	Sigma22LEB [Pa]	Tau12LEB [Pa]	Plate Slend Req	Plate Aspect Ratio	Flange Slend Req
	Web Slend Req	Web Flange Slend Req	Sec Flange Slend Req	Sec Web Slend Req	Sec Stiff Aspect Ratio	PULS Error Code		
Pl2	LC05_Double_Mom	0.50	Failed(uf)	1.23	etaBS/allowBS	Geom OK	SP Buckling	Cm_TWF67.run(1)
	2.37697	0.694562	0.009	5	F	C	0.175	0.008
	0	0	0	0	2.06e+011	0.3	2.35e+008	2.35e+008
	7.07358e+007	3.97818e+007	3.97818e+007	-3.7552e+007	0	Int	9.16001e+007	5.15159e+007
	5.15159e+007	-4.86283e+007	0.772	1.000	OK	5.74001e+007	3.22818e+007	3.22818e+007
	-3.04723e+007	1.232	1.000	Failed	3	1.49266e+008	8.3947e+007	8.3947e+007
	-7.92416e+007	5.74001e+007	3.22818e+007	3.22818e+007	-3.04723e+007	Passed	Passed	N/A
	Passed	N/A	N/A	N/A	N/A	0		
Pl3	LC05_Double_Mom	0.50	Failed(uf)	1.83	etaBS/allowBS	Geom OK	SP Buckling	Cm_TWF67.run(1)
	2.41213	0.718497	0.009	5	F	C	0.175	0.008
	0	0	0	0	2.06e+011	0.3	2.35e+008	2.35e+008
	5.36885e+007	6.03628e+007	6.03628e+007	-1.02015e+007	0	Int	5.18407e+007	5.82854e+007
	5.82854e+007	-9.85042e+006	1.036	1.000	Failed	2.92898e+007	3.2931e+007	3.2931e+007
	-5.56545e+006	1.833	1.000	Failed	1	7.70463e+007	8.66244e+007	8.66244e+007
	-1.46398e+007	2.92898e+007	3.2931e+007	3.2931e+007	-5.56545e+006	Passed	Passed	N/A
	Passed	N/A	N/A	N/A	N/A	0		

CSR Tank code check results report – Stiffened and unstiffened panels.

The CSR Tank buckling is based on use of the PULS program for buckling and ultimate strength capacity assessments of stiffened and unstiffened panels based on non-linear large deflection plate theory.

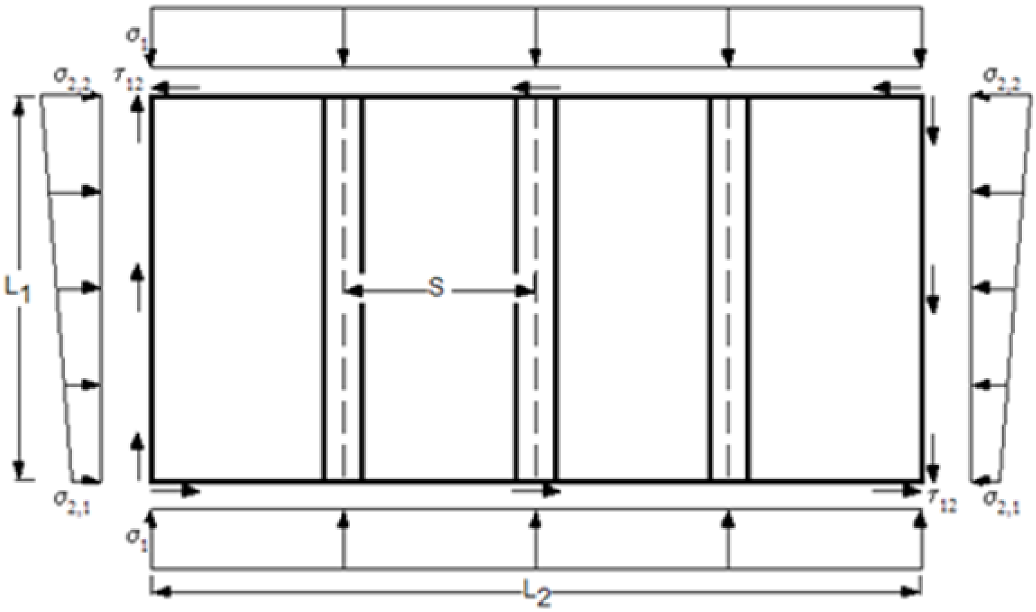
For further details, we refer to the PULS Manuals - Buckling package documentation.

Execution of the PULS program requires an installation of the Nauticus Hull FEM package.

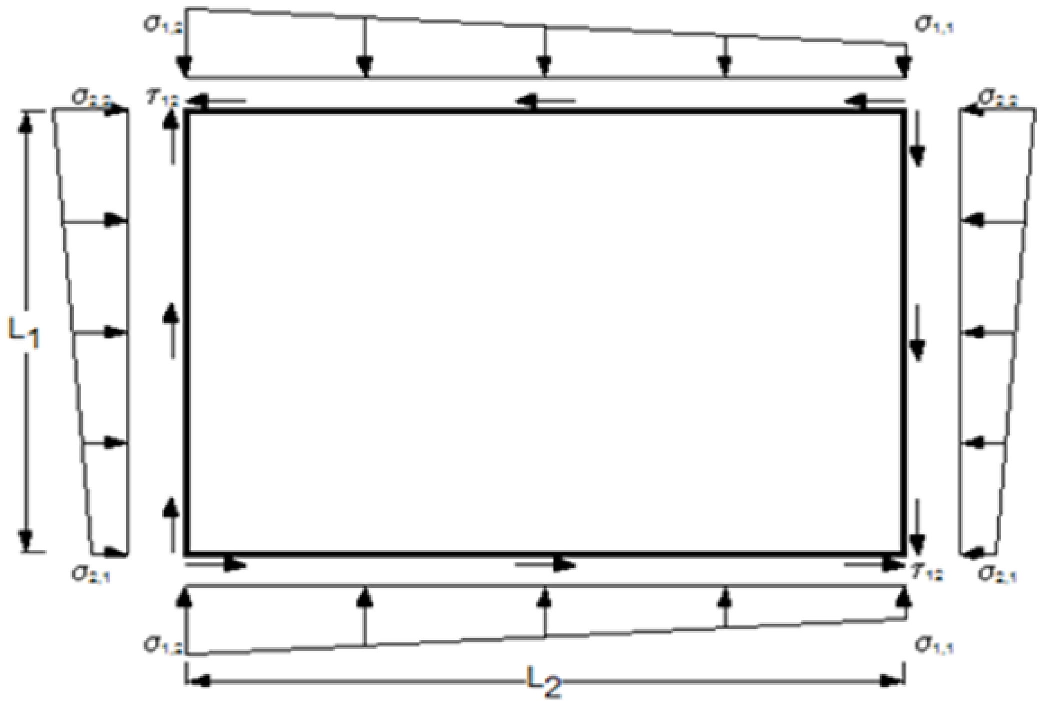
Panel	The name of the panel
Loadcase	The name of the loadcase
Position	Centroid of panel. 0.50 is on the middle of panel. Not important for CSR Tank
Status	Status of a panel is either: *OK*, *Failed (Uf)* or *Failed (geo)*
UfTot	Utilisation factor
Formula	The formula that is governing
GeomCheck	Shows whether the panel's geometry is consistent with regards to the code check
SubCheck	Which subcheck is governing
Run	The name of the run
etaUC/allowUC	Utilisation UC (Ultimate Capacity)
etaBS/allowBS	Utilisation BS (Buckling Strength)
etaLEB/allowLEB	Utilisation LEB (Local, Linear Elastic Buckling)
etaGEB/allowGEB	Utilisation GEB (Global, Linear Elastic Buckling) **(Stiffened panel only)**
Panel Length	
Panel Width/Stiffener spacing	
Thickness	Plate thickness (net)
No Primary Stiffeners	Number of primary stiffeners **(Stiffened panel only)**
Stiffener Type	F=Flatbar, L=Angle, T=Tee
Stiffener Boundary	S=Sniped, C=Continous
Stiffener Height	Stiffener Height
Web Thickness	Web Thickness (net)
Flange Width	Flange Width
Flange Thickness	Flange Thickness (net)
Tilt Angle	Not used in Genie

Stiffened panel:

E-Modulus	
Poisson's Ratio	
Yield Stress Plate	
Yield Stress Stiffener	
In Plane Support	Int=Integrated Panel, GL=Left/right edges free, GS=Upper/lower edges free
Support Left	***Rotational support - SS=Simply Supported, CL=Clamped (Unstiffened panel only)***
Support Right	
Support Upper	
Support Lower	
Sigma11	Axial stress - Stress in X direction along short side, *see illustration*
Sigma12	Axial stress - Stress in X direction along short side, *see illustration*
Sigma21	Transverse stress - Stress in Y direction along long side, *see illustration*
Sigma22	Transverse stress - Stress in Y direction along long side, *see illustration*
Tau12	Shear stress in panel
Pressure	Lateral pressure on panel
Sigma11UC …	Value of Sigma11 when UC is reached
Tau12UC	Value of Tau12 when UC is reached
etaUC	Actual usage factor for UC
allowUC	Allowable usage factor for UC
Sigma11BS …	Value of Sigma11 when BS is reached
Tau12BS	Value of Tau12 when BS is reached
etaBS	Actual usage factor for BS
allowBS	Allowable usage factor for BS
Sigma11LEB …	Value of Sigma11 when LEB is reached
Tau12LEB	Value of Tau12 when LEB is reached
etaLEB	Actual usage factor for LEB
allowLEB	Allowable usage factor for LEB
Sigma11GEB …	**Value of Sigma11 when GEB is reached**
Tau12GEB	**Value of Tau12 when GEB is reached**
etaGEB	**Actual usage factor for GEB**
allowGEB	**Allowable usage factor for GEB**



Unstiffened panel:



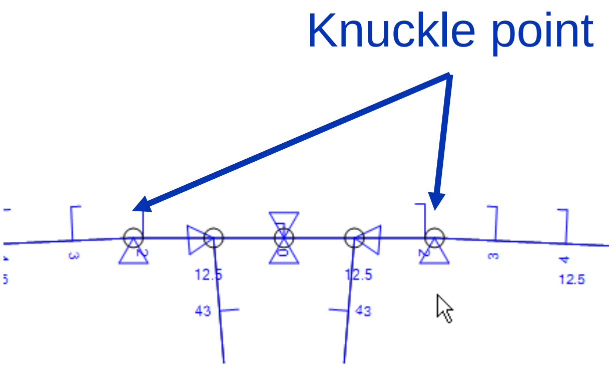
The left, right, upper and lower input of the boundary conditions corresponds to the sides of these illustrations.

4.9 Limitations

4.9.1 Panel split along a knuckle line

The structural elements have to be planar, so if a region between plates is piecewise planar but not globally planar, then it necessarily is split into separate structural elements.

If a stiffened panel contains a knuckle line, like for instance in the strength deck, the stiffened panel will be split into two panels along the knuckle line.



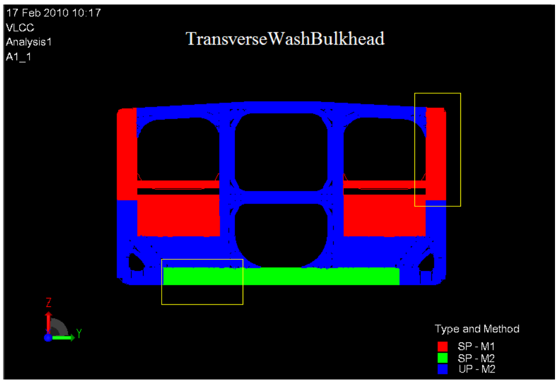
4.9.2 Transverse wash bulkheads

A workaround to solve the “assignment of method” problem for the transverse wash bulkhead

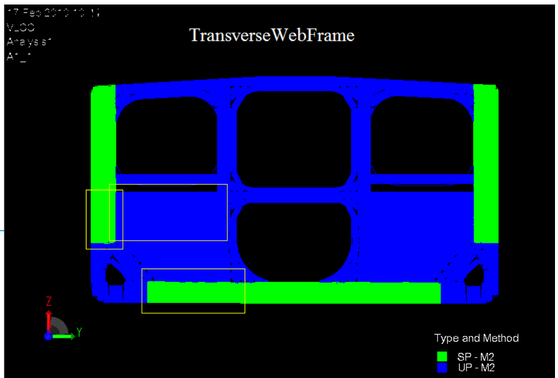
The panels of a Wash Bulkhead should be created, by assigning different structural types to it.

The following text and illustrations explain how this should be done. The areas in yellow squares are the erroneous ones.

If we assign to it the structural type TransverseWashBulkhead we get:

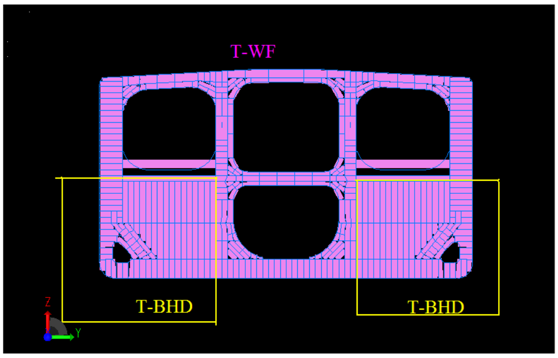


If we assign to it the structural type TransverseWebFrame (default) then we get:

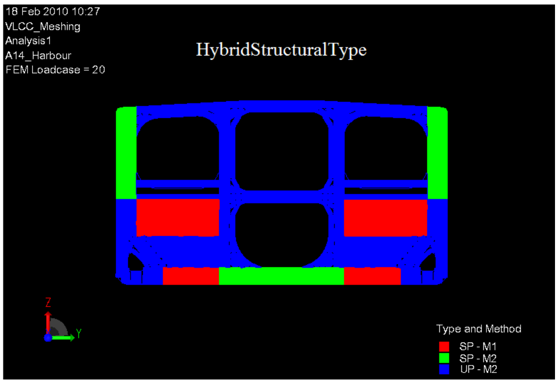


Clearly, the errors in all cases can be explained with the aid of the illustration to the right.

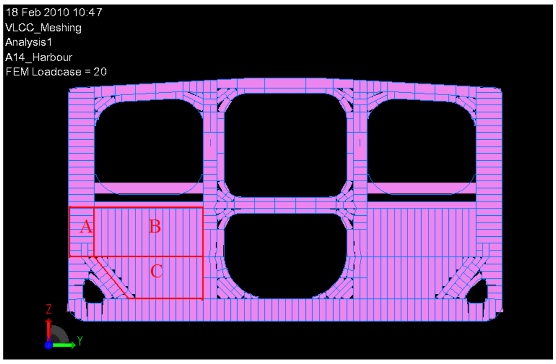
The areas inside the yellow squares have to be treated as transverse bulkheads, while the rest of the structure has to be treated as transverse web frame. If this happens, then the cases 2 and 3 show clearly that all the results satisfy the requirements of figure D.5.4 of the CSR.



In view of this observation we assign these two structural types to the plates of the above areas:



Apart from the above, the illustration to the right explains why some of the panels of the structural elements A and C are UP-M2 and not SP-M1; simply because they are not regularly stiffened (as opposed to B).



5 HOW TO DO PANEL CODE CHECKING – DNV GL OR CSR BC & OT

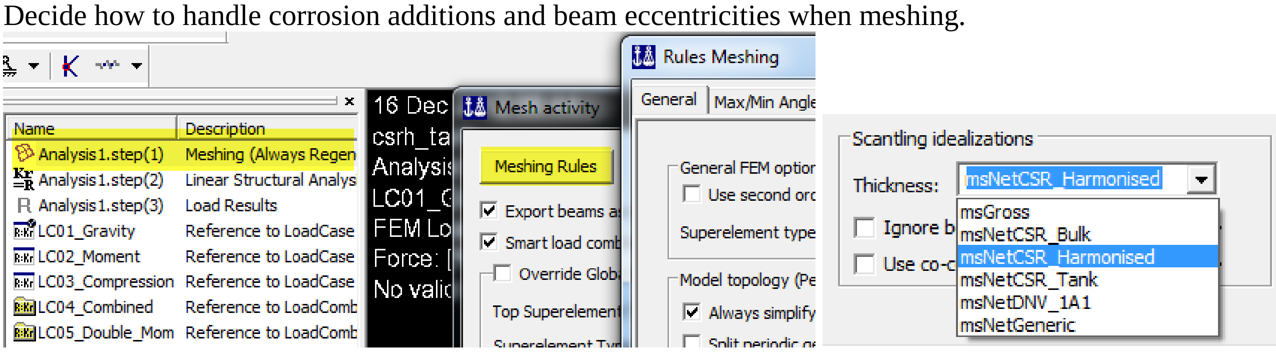
A reference case will be used; the slice of a bulk carrier that was introduced earlier in this user documentation.

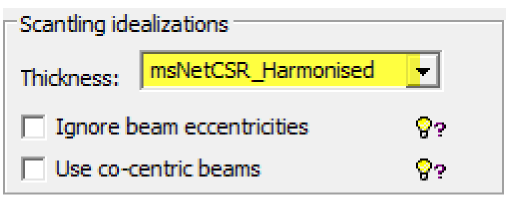
The following procedure may be used when performing code checks; each is described in detail except for the three first steps (please consult the User Manual Vol. I for guidance).

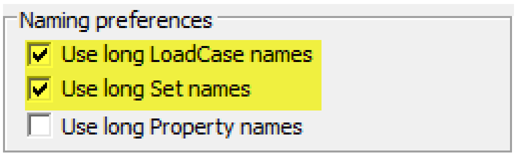
- Make a plate model. Beams may be included
- Decide how to handle corrosion additions when meshing.
- Run the finite element analysis
- Define relevant load combinations if they were not part of the above analysis.
- Create a capacity manager
- Define the panels
- Create a code check run
- Assign specific settings to individual panels
- Compute the code checking forces
- Perform the code check and investigate the results graphically or from the browser.
- If necessary modify plate thickness, materials or other code checking parameters. Note that changes made to structure model requires that you re-run the FEM analysis and re-generate the panels. Local changes on some of the capacity panels are allowed, but these will be lost when re-creating the panels.
- View and / or print results

5.1 Preparations for meshing

Decide how to handle corrosion additions and beam eccentricities when meshing.



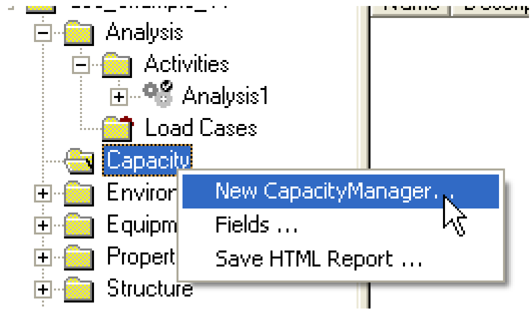


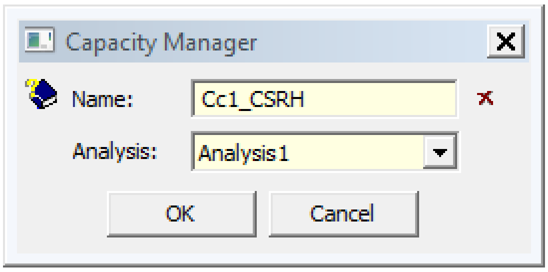


5.2 Create a capacity manager

The purpose of a capacity manager is to decide which analysis results to use in the code checking. It is possible to have several analysis activities in GeniE where you can have different load cases as well as analyse subsets of your model. To be able to do code checking you need to define multiple capacity managers referring to the different analysis in question.

In this case we have one analysis, and one capacity manager is created from the browser.

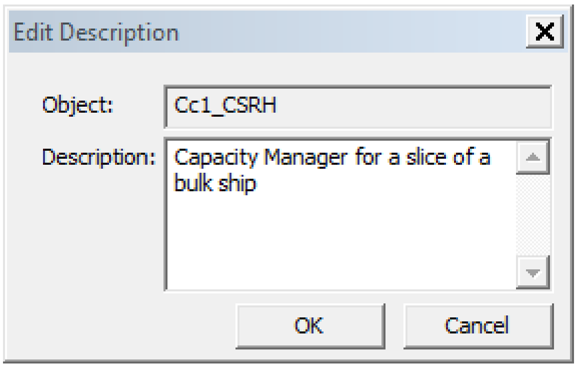


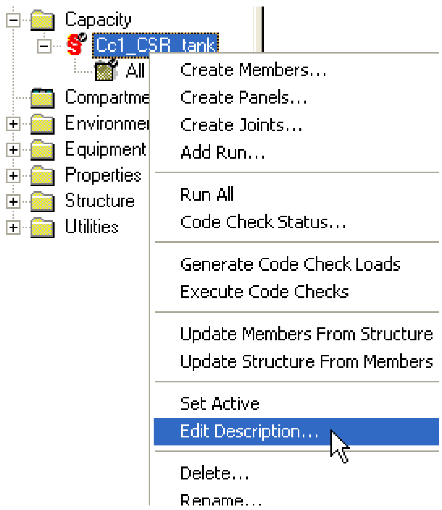


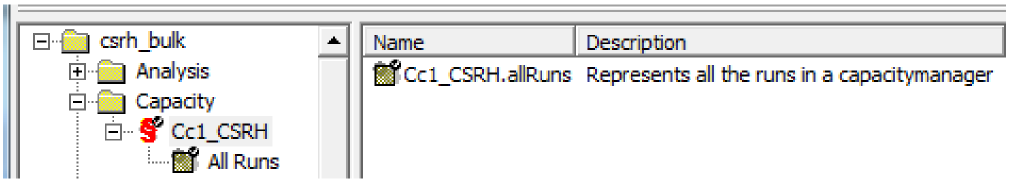
Note that when naming the capacity manager it must be given a unique name. You cannot give it the same name as a set for instance.

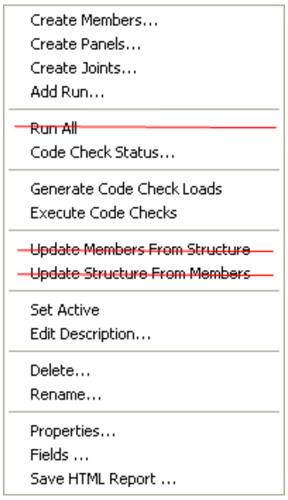
You may add additional description to the capacity manager. The description is also shown in the browser.

When making a report, the descriptions are also documented.






csrh_bulk
Analysis
Capacity
Cc1_CSRH
All Runs
Name
Description
Cc1_CSRH.allRuns
Represents all the runs in a capacitymanager



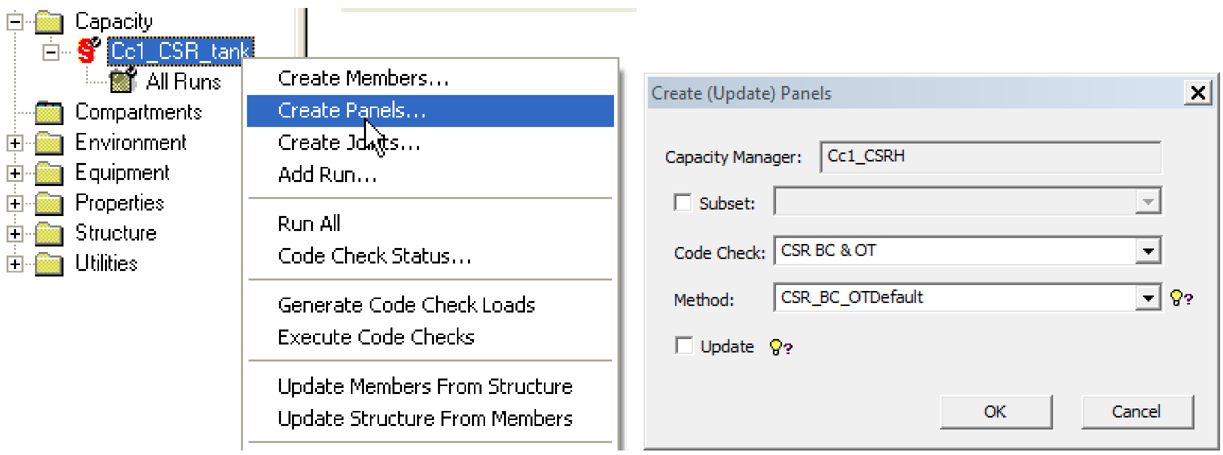
IMPORTANT LIMITATION: The menu below is common to all code checks. However the menu items *Run All*, *Update Members From Structure* and *Update Structure From Members* are intended to be used when using redesign for a member code check (beams).

The menu items *Run All*, *Update Members From Structure* and *Update Structure From Members* should not be used for a CSR BC & OT or DNV GL rules run.

5.3 Defining panels

When modelling a concept model it is possible to make continuous plates that span several beams. This means that the concept model is different from a panel capacity model which spans between two beams only. It is therefore necessary to split up the concept model into elementary panel capacity models.

When the model is split or kept, the default buckling lengths are set since they are the same as the length or the breadth of a capacity panel.



If you are working on a large bulk carrier or VLCC it is necessary to divide the model into smaller subsets and create panels for one subset at a time. This can be done by checking the *Subset* check button and choose a subset from the subset list. In this case we have chosen to create panels for the whole model.

When creating panels there are 3 choices available:

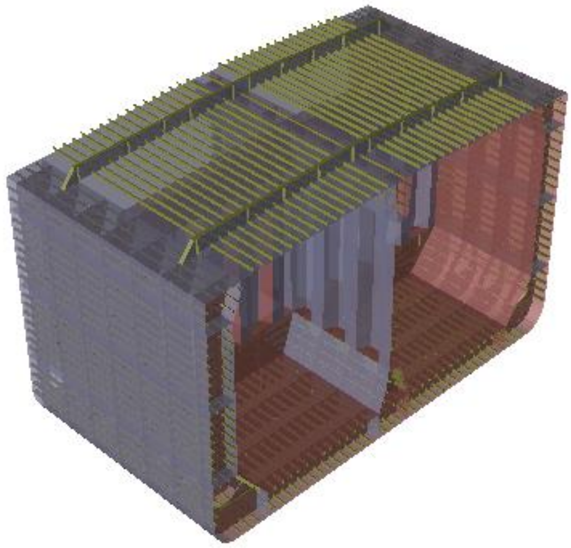
- **CSR_BC_OTDefault** : Is the algorithm to be used when preparing a CSR Harmonised code check. Will create planar panels and also planer panels for regions with large curvature. (Ref. Pt.1 Ch.8 Sec.4 §2)
- **CSR_BC_OTIrregularPanel** - same geometry as above, but with different stress calculation. (Ref. Pt.1 Ch.8 Sec.4 §2.3 and App.1 §2.2).
- **CSR_BC_OTOnePanelPerThickness** - Alternative method, specially intended for displaying more detailed result for large corrugated bulkhead. (Ref. Pt.1 Ch.8 Sec.4 §3.4.1).

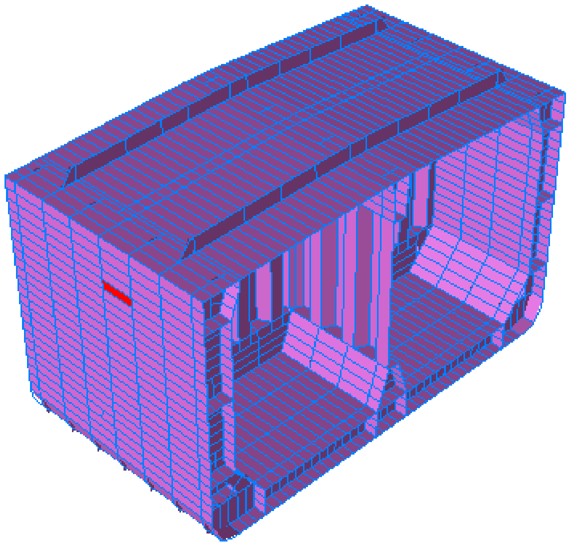
Note that the **CSR_BC_OTDefault** method also will create panels for curved regions. However, this method should not be used for subsequent execution with the CSR Bulk or CSR Tank code-check, since these code-checks in GeniE do not include specific computations for panels created from curved regions. If however, this method is used with CSR Bulk or CSR Tank code-check, the validity of the results must be evaluated by the user – the more panel curvature increases the more non-consistent results.

The other panel creation methods will only accept slightly curved regions, within 1 degree of curvature, and only create planar panels.

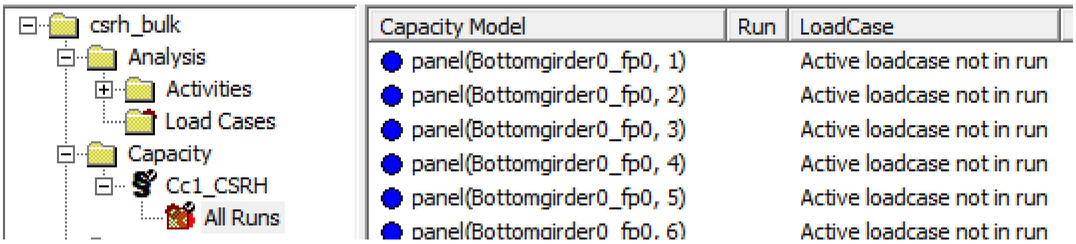
The naming convention of capacity panels refer to the plates. If Plate *Pl20* is split into two capacity panels they are denoted *panel(Pl20,1)* and *panel(Pl20,2)*.

The pictures show the concept model and the capacity panels.





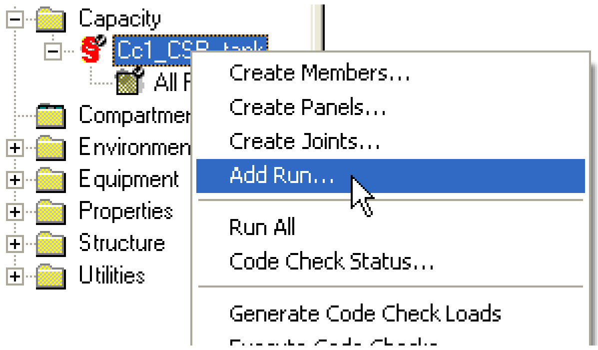
The browser lists each capacity panel. If you have several capacity managers you need to specify which manager is active (select a manager, RMB and choose *Set Active*). In this case “Cc1_CSRH” is the only capacity manager and it is set to active.



5.4 Create a DNV GL or CSR BC & OT code check run

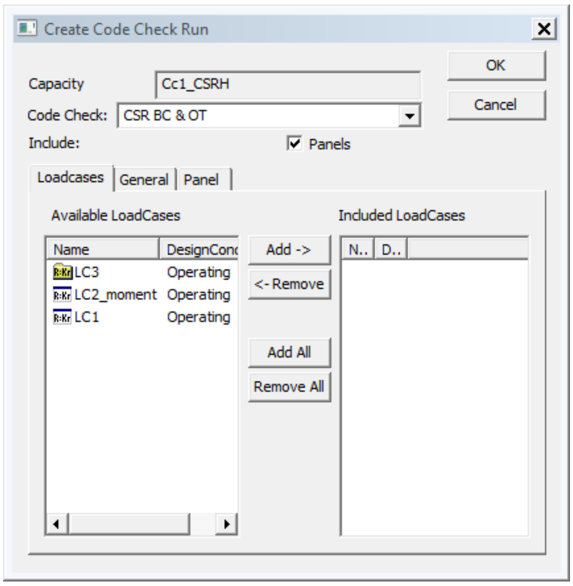
During the definition of code check runs you decide

- which code of practice to use
- which load cases to use
- global code checking parameters (i.e. those who apply to the entire capacity model), for example code check settings, safety factors.

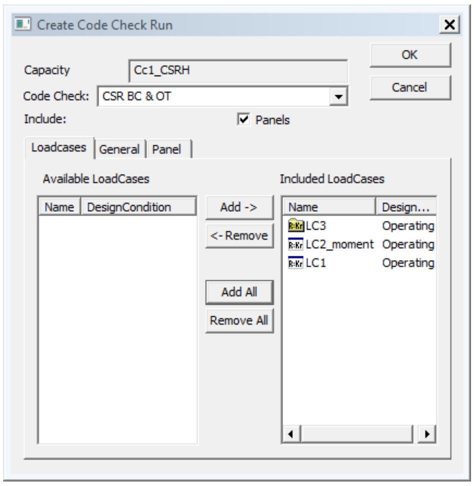


The code check run is added from the browser.

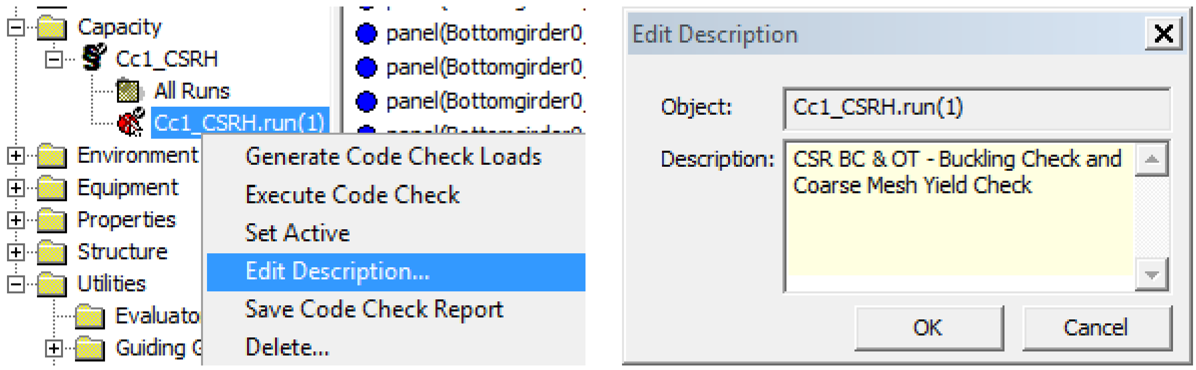
Note that from V7.3 the user must select either to make a run of type "DNV GL rules" or "CSR BC & OT", since there are some differences between the 2 code checks.



Furthermore, all or a selection of the load cases have to be added to the CSR BC & OT code check run.

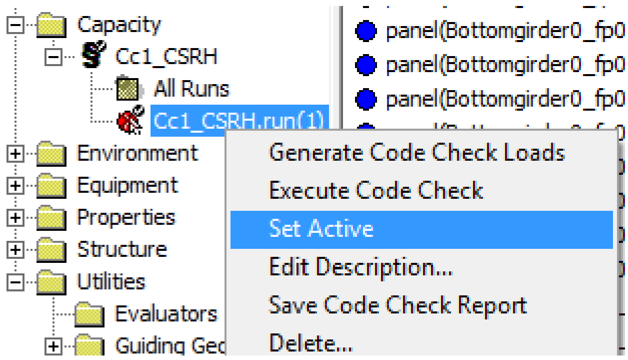


You may modify the code check run from the browser and define additional information to the run.



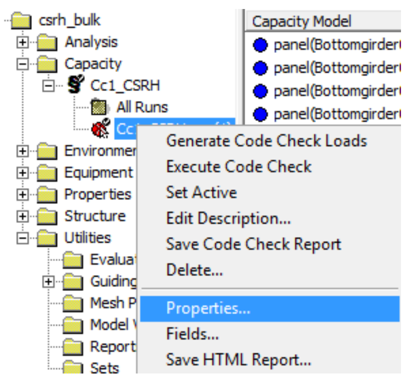
As default,the description it will show type of run with current settings. However, if the user enter another text, then that will have higher priority,

The same procedure may be used to create another code check run. You then have to specify which one is active from the browser.



5.4.1 Define global general code checking parameters

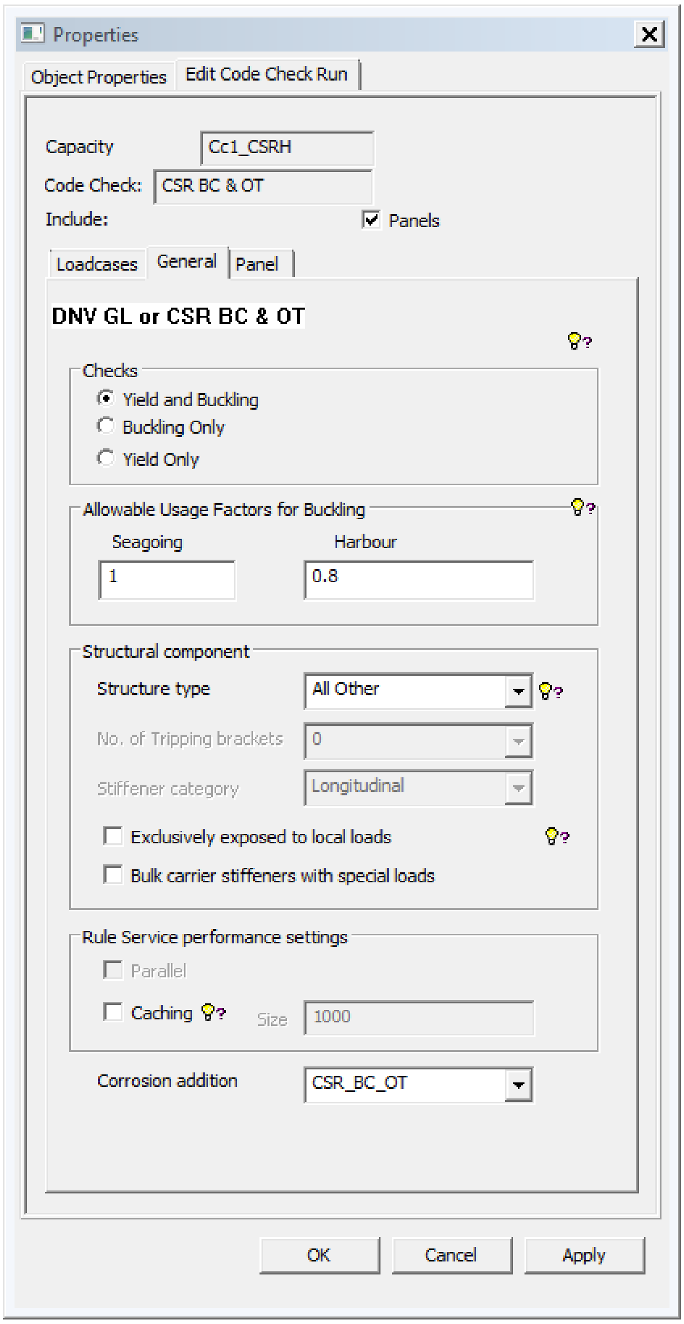
You specify and modify the general code checking parameters when you define the code check run, or you may modify at a later stage. To modify, select *Properties* as shown on the picture to the right.



For a further description of parameters shown herein, please consult the relevant codes of practices. The general code checking parameters are global, i.e. they apply to all capacity panels.

5.4.1.1 *General parameters*

The general parameters for the CSR BC &OT run are shown below.



Checks

The user can choose to perform code checks for buckling, yield or both at the same time.

Allowable Usage Factors

The Allowable Usage Factors for buckling are user defined and represent the usage factors considered on operation condition (**Seagoing** and **Harbour**). The definition of these values will set the threshold values for the code check result values.

Structure type for C_y factor

Structural component

- Structure type for calculation of Flong factor:
- For vertically stiffened single side skin of bulk carrier, Cy is calculated according to Table 2 by using c1 as given in §2.2.3
- For corrugation of corrugated bulkheads, Cy is calculated according to Table 2 by using c1 as given in §2.2.3
- For other structure types, Cy is calculated according to Table 2 by using c1 as given for Method A or B in §2.2.3.

Stiffener category

The alternative Longitudinal Stiffener should always be used in FEM.

Exclusively exposed to local loads

Setting options for Safety factor

- Used to set default Safety factor options for plate and stiffeners. See rule text.

Bulk carrier stiffeners with special loads

Corrosion addition

The Corrosion Addition Rule may be set on the General tab of the capacity run of type *DNV GL or CSR BC & OT* and *CSR-Tank*.

- For a CSR BC & OT run the user should select *CSR BC&OT* for Corrosion Addition rule (default).
- For a DNV GL run the user should select *DNVGL* for Corrosion Addition rule.
- For a DNV GL run using PULS (the user selects run type *CSRTank*) and the user should select *DNVGL* for Corrosion Addition rule.

Note: At this moment, the default value is NOT picked up from selected Corrosion Addition rule on the Edit | Rules | Meshing.

Further, the panels must be re-created after modifying the corr add rule.

5.4.2 Define global panel parameters

The global panel parameters (those who apply to the all capacity panels) may be changed from default values when you define the code check run or later. The default values are shown in the following for the CSR BC & OT or DNV GL rules code check alternative.

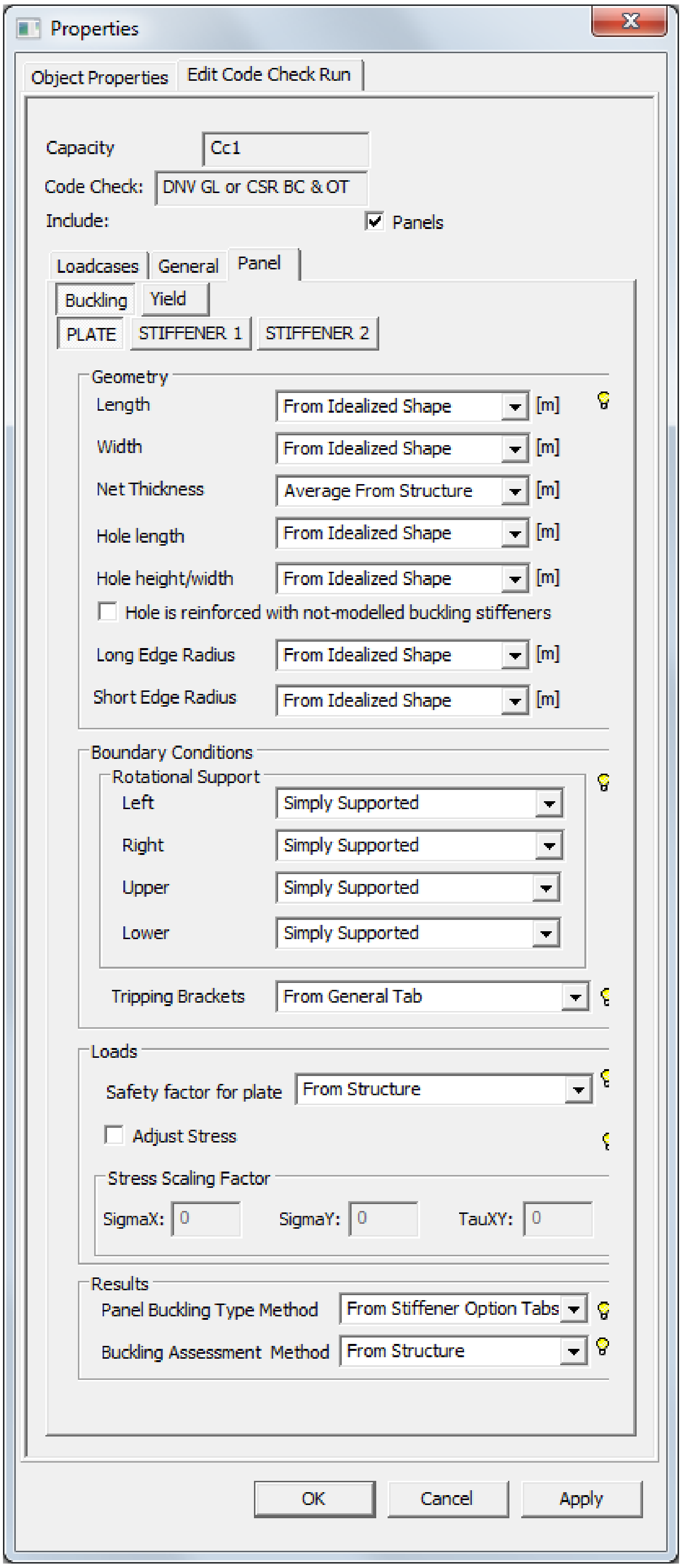
- **CSR BC&OT Default** Is the default method for generating panels for CSR BC&OT
- **DNV GL rules Default** Is the default method for generating panels for DNV GL rules

Panel properties tabs :

- **Buckling** tab gives the user control over the buckling check type to be performed and detailed settings for individual panels.
- **Yield** tab gives the user control over the yield check type to be performed and detailed settings for individual panels.

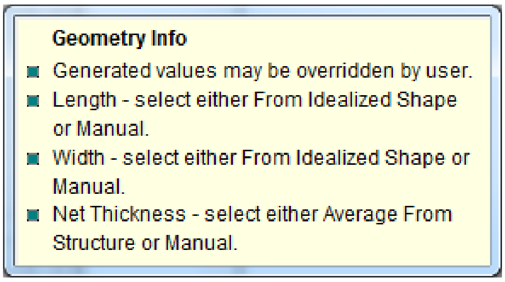
5.4.2.1 *Panel parameters*

The default panel options for unstiffened panels are shown below. You can define if the panel has a stiffener at one or both sides, or no stiffening at all.

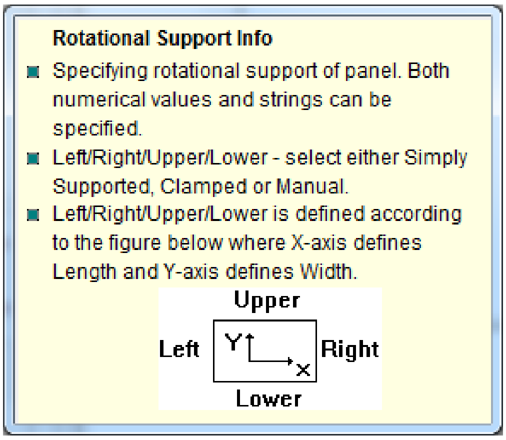


PLATES

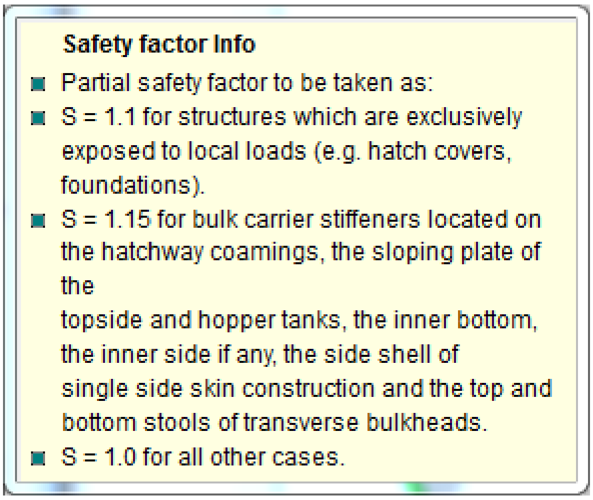
Geometry



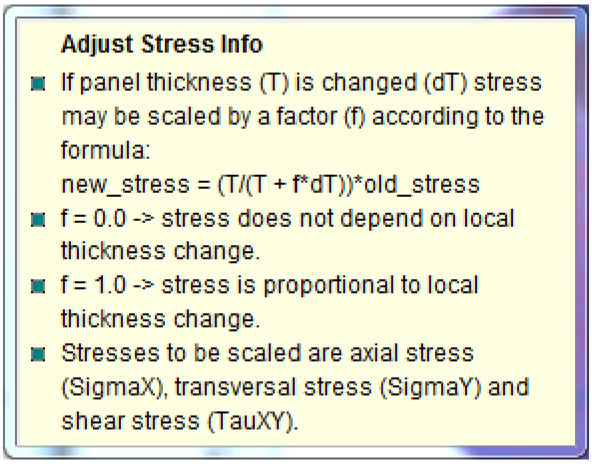
Boundary conditions



Loads



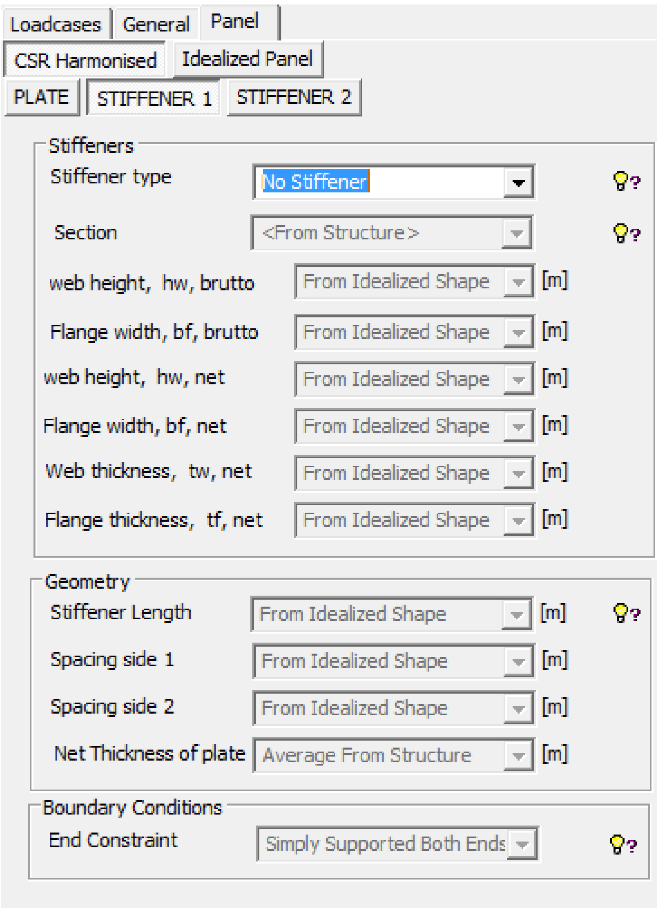
Stress scaling



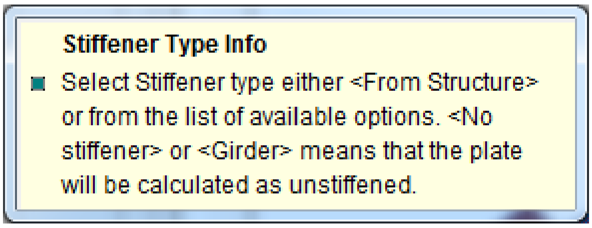
Adjust Stress Info

- If panel thickness (T) is changed (dT) stress may be scaled by a factor (f) according to the formula:
  new_stress = (T/(T + f*dT))*old_stress
- 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).

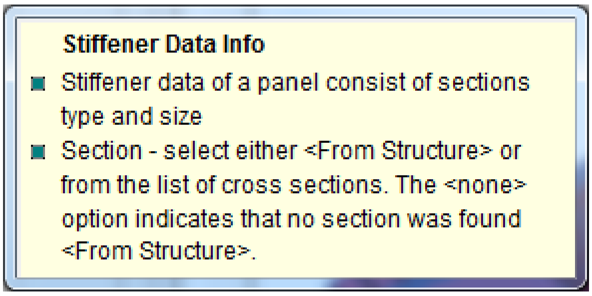
STIFFENERS



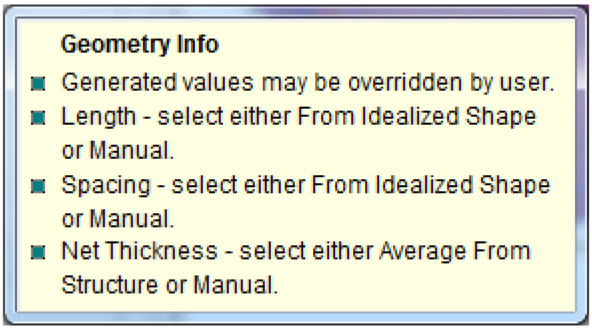
Loadcases
General
Panel
CSR Harmonised
Idealized Panel
PLATE
STIFFENER 1
STIFFENER 2
Stiffeners
Stiffener type
No Stiffener
Section
<From Structure>
web height, hw, brutto
From Idealized Shape
[m]
Flange width, bf, brutto
From Idealized Shape
[m]
web height, hw, net
From Idealized Shape
[m]
Flange width, bf, net
From Idealized Shape
[m]
Web thickness, tw, net
From Idealized Shape
[m]
Flange thickness, tf, net
From Idealized Shape
[m]
Geometry
Stiffener Length
From Idealized Shape
[m]
Spacing side 1
From Idealized Shape
[m]
Spacing side 2
From Idealized Shape
[m]
Net Thickness of plate
Average From Structure
[m]
Boundary Conditions
End Constraint
Simply Supported Both Ends



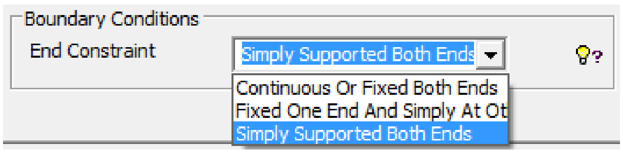
Stiffener Type Info
Select Stiffener type either <From Structure> or from the list of available options. <No stiffener> or <Girder> means that the plate will be calculated as unstiffened.



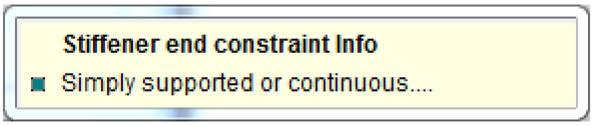
Stiffener Data Info
Stiffener data of a panel consist of sections type and size
Section - select either <From Structure> or from the list of cross sections. The <none> option indicates that no section was found <From Structure>.



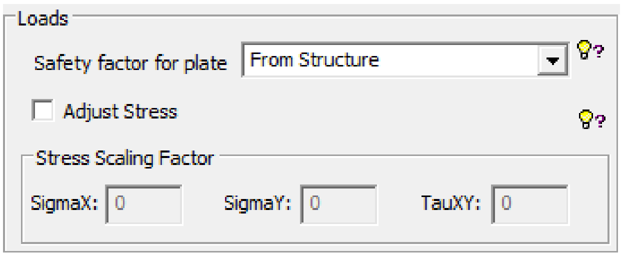
Geometry Info
Generated values may be overridden by user.
Length - select either From Idealized Shape or Manual.
Spacing - select either From Idealized Shape or Manual.
Net Thickness - select either Average From Structure or Manual.



Boundary Conditions
End Constraint
Simply Supported Both Ends
Continuous Or Fixed Both Ends
Fixed One End And Simply At Ot
Simply Supported Both Ends



Stiffener end constraint Info
Simply supported or continuous....



Loads
Safety factor for plate
From Structure
Adjust Stress
Stress Scaling Factor
SigmaX:
0
SigmaY:
0
TauXY:
0

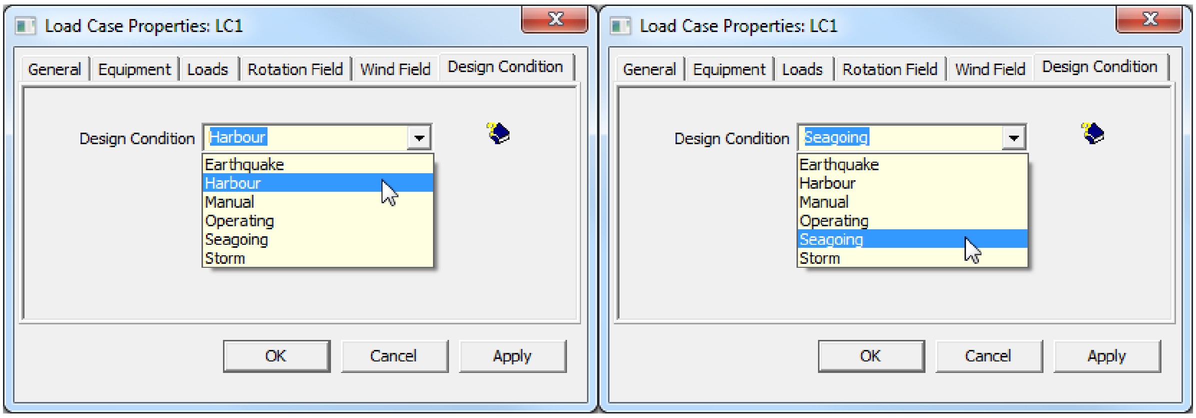
5.4.2.2 *Yield check parameters*

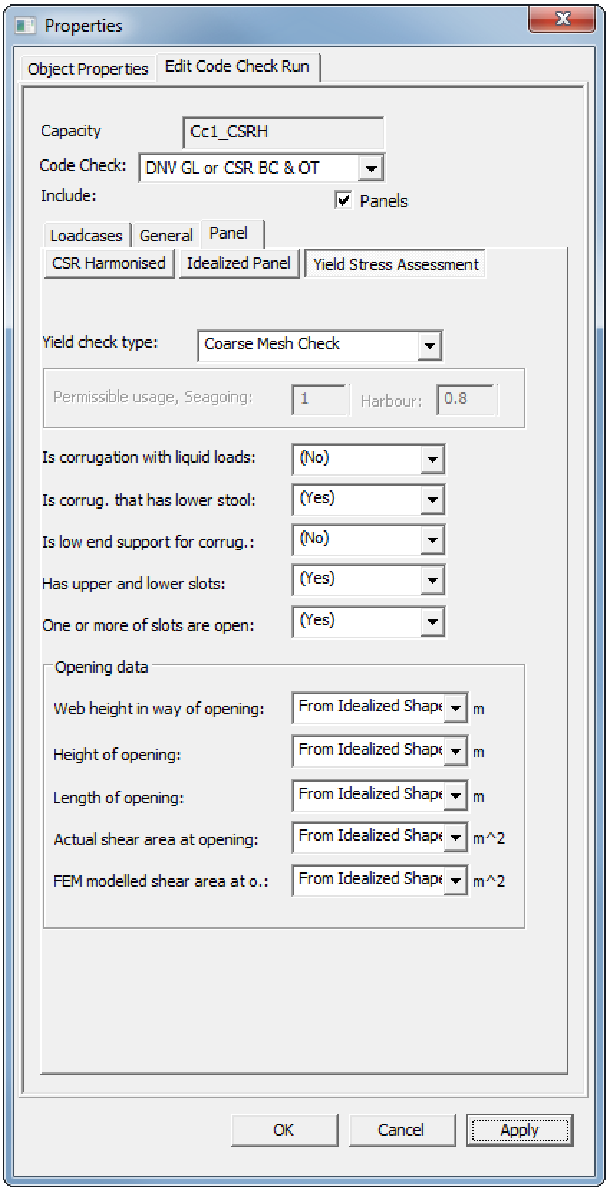
The yield stress assessment is comparing the average membrane von Mises stress per element in the panel towards material yield stress and the permissible yield usage as is defined in the rules or manually by the user.

If yield code checking has been included in the *General* tab options, the user can now adjust the settings for the yield stress assessment. Note that the Run has to be added (by clicking OK) before it is possible to access these settings (via right-click on the run > properties).

Design Condition

Note that for both “Coarse Mesh Check” and “Manual Check”, the permissible usage factors refer to the Design Condition that is set in the investigated load case:





Yield check type

The user can choose between *Coarse Mesh Check* and *Manual Check*.

Permissible usage

The permissible usage is depending on the load case and structural component type.

When selecting "Manual Check" in the yield check type selection, the permissible usage factors might be specified manually. Seagoing input is for load combination "S + D" (static and dynamic) and Harbour input is for load combination "S" (static).

For "Coarse Mesh Check" selection, the permissible usage factors are taken from Table 10 (see below) in the rules [5.2.3], including special cases and other requirements as specified in [5.2.5], [5.2.6] and [5.2.7.]

The program will try to identify as much of the special cases and other required input for these as possible based on other available inputs. But the user will have to verify that the default values shown in parentheses are correct and update accordingly if not.

Important: you may update settings in the Run-properties if the attribute is common for all panels. Alternatively, select one or several panels to update these individually.

Depending on selections, other irrelevant input fields will be disabled.

Default values are shown with parentheses around it, while user inputs are shown without parentheses.

The first "Is corrugation with liquid loads:", "Is corrug. that has lower stool:" and "Is low end support for corrug.:" are related to the options in table 10 (see below).

The "Opening data" input is related to [5.2.6], Shear stress correction for cut-outs and is described in the rules.

"Web height in way of opening" (h): Height of web of girder, in mm, in way of opening, see Table 1. Where the geometry of the opening is modelled, h is to be taken as the height of web of the girder deducting the height of the modelled opening.

"Actual shear area of opening" (Ashr-n50): Effective net shear area of web, in mm2, taken as the web area deducting the area lost of all openings, including slots for stiffeners, calculated in accordance with Ch.3, Sec.7, [1.4.8].

"FEM modelled shear area at o." (Ashr-n50): Effective net shear area of the web, in mm2, taken as the web area without the all opening areas and without the slots for stiffeners, in accordance with Ch.3, Sec.7, [1.4.8].

The "Has upper and lower slots" and "One or more of slots are open" are related to options shown in table 11. "Exceptions for shear stress correction" in [5.2.7].

Table 10 : Coarse mesh permissible yield utilisation factor:

Structural component	Coarse mesh permissible yield utilisation factor, λ_{yperm}
Plating of all longitudinal hull girder structural members, primary supporting structural members and bulkheads. Face plate of primary supporting members modelled using shell or rod elements. Dummy rod of corrugated bulkhead	1.0 (load combination S+D) 0.8 (load combination S)
Corrugation of vertically corrugated bulkheads with lower stool and horizontally corrugated bulkhead, under lateral pressure from liquid loads, for shell elements only. Supporting structure in way of lower end of corrugated bulkheads without lower stool [1].	0.90 (load combination S+D) 0.72 (load combination S)
Corrugation of vertically corrugated bulkheads without lower stool under lateral pressure from liquid loads and without lower stool, for shell elements only.	0.81 (load combination S+D) 0.65 (load combination S)

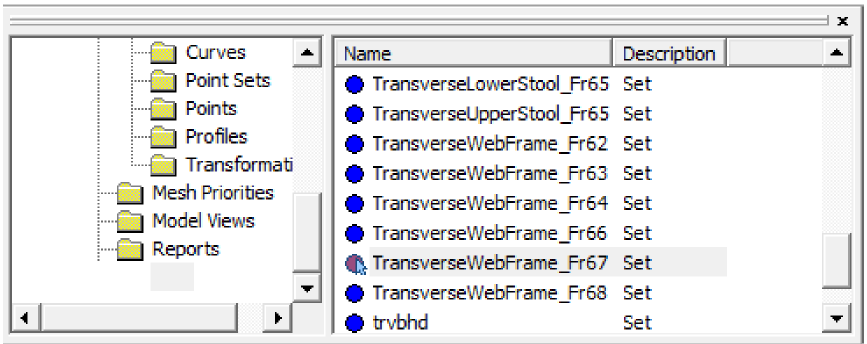
NOTES related to yield check:

1. Currently, only shell elements are checked.
2. Every shell element within the panels is checked, independently of the element size. (No averaging of small elements is done).
3. The material yield strength from the FEM model is used.
4. The element thickness from the FEM model is used (Net thickness)

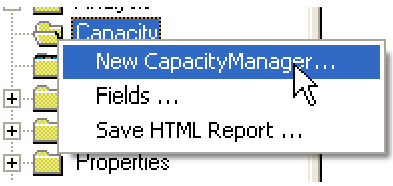
5.5 Make a capacity manager and a run for one specific set and performing the code check.

When working with CSR BC & OT on large models it is generally impractical to work with the entire model in one capacity manager. To keep an overview and for calculations not to be too time consuming, you should make a separate capacity manager with one run for each set of interest.

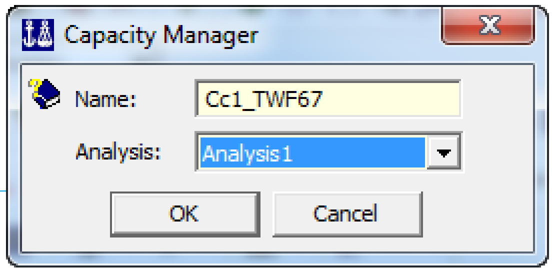
By looking at the list of Sets in the browser we find a set of interest, for instance *TransverseWebFrane_FR67*.



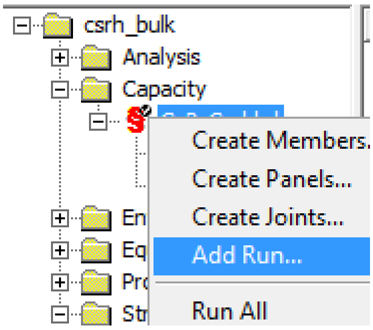
You make a new Capacitymanager and give it a name that easily identifies which set is going to be handled in this capacitymanager.

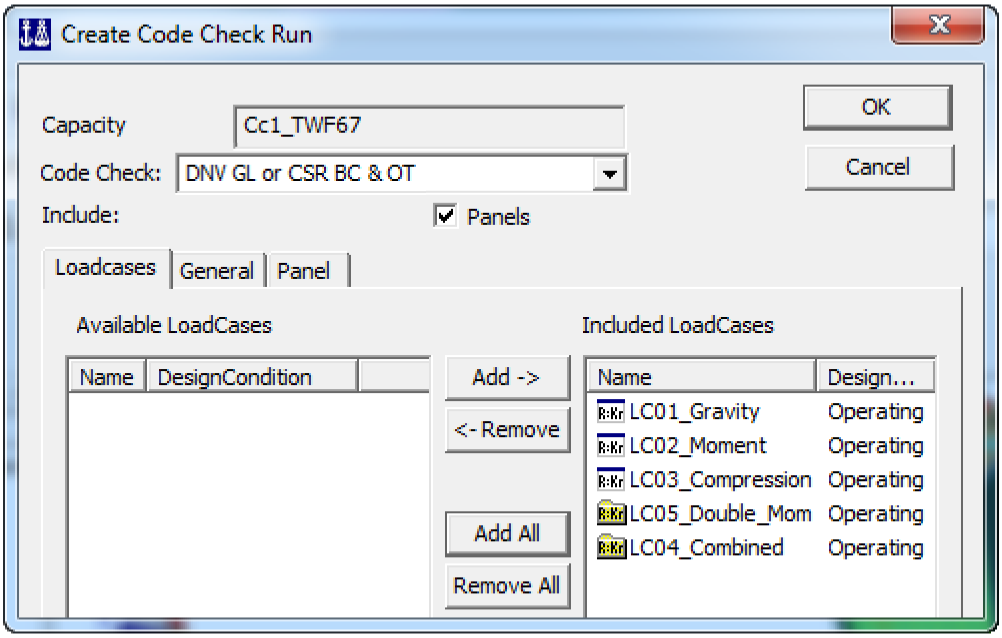


n/software

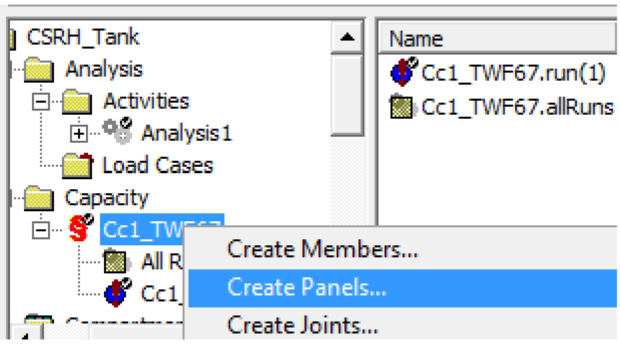


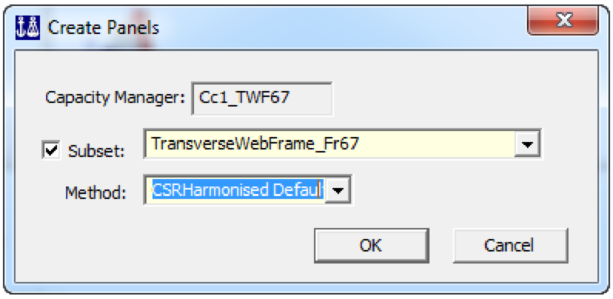
You add a run, choose DNV GL or CSR BC & OT as code check and include the loadcases of interest. In the illustration all loadcases have been selected.



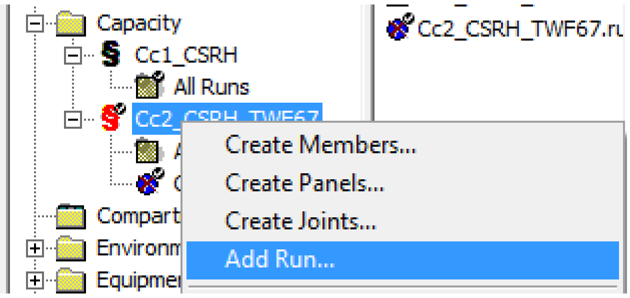


Select *Create Panels* from the **RMB** menu and make sure the Subset checkbutton is ticked off and that your set of interest is selected. Also make sure to select CSR BC & OT Default in the Method drop down menu.

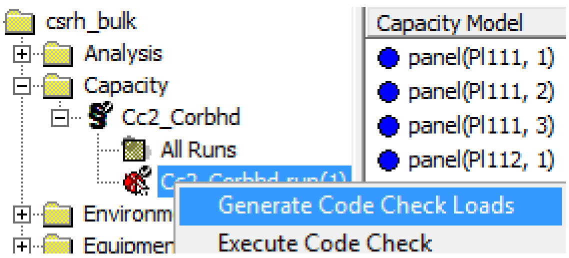


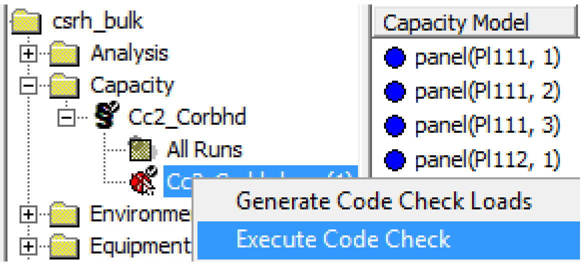


Select the Capacity manager, RMB and Add run.

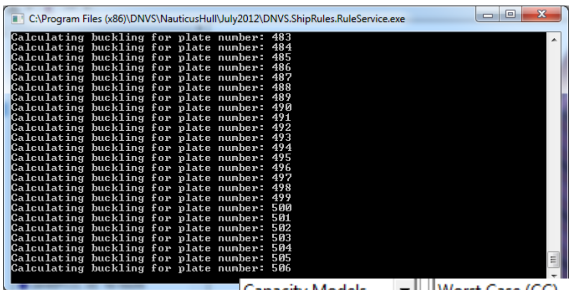


RMB click on your run, select *Generate Code Check Loads* and then select *Execute Code Check.*





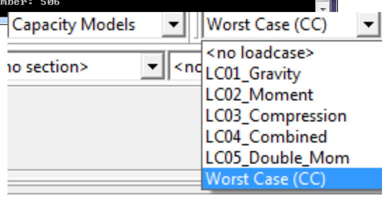
When you have selected *Execute Code Check* (for a CSR BC & OT run) the RuleService starts running in a separate console window, similar to the one to the right. Observe that the calculation is running as long as new numbers keep showing up from the bottom of the console window.

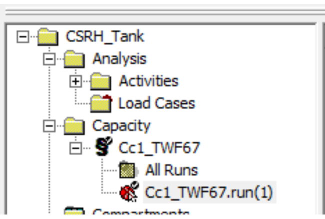


Running the codecheck is often quite time consuming.

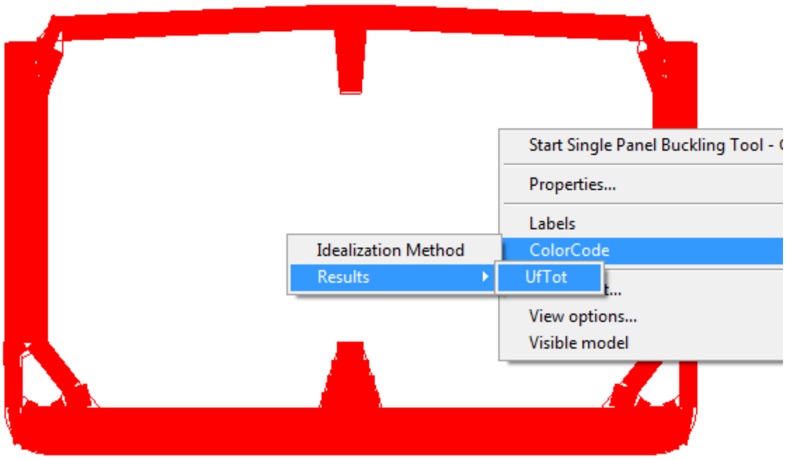
The console window closes itself and disappears when the calculation is completed.

The results are now available in the browser after you have selected a specific Resultcase, or the worst Resultcase:

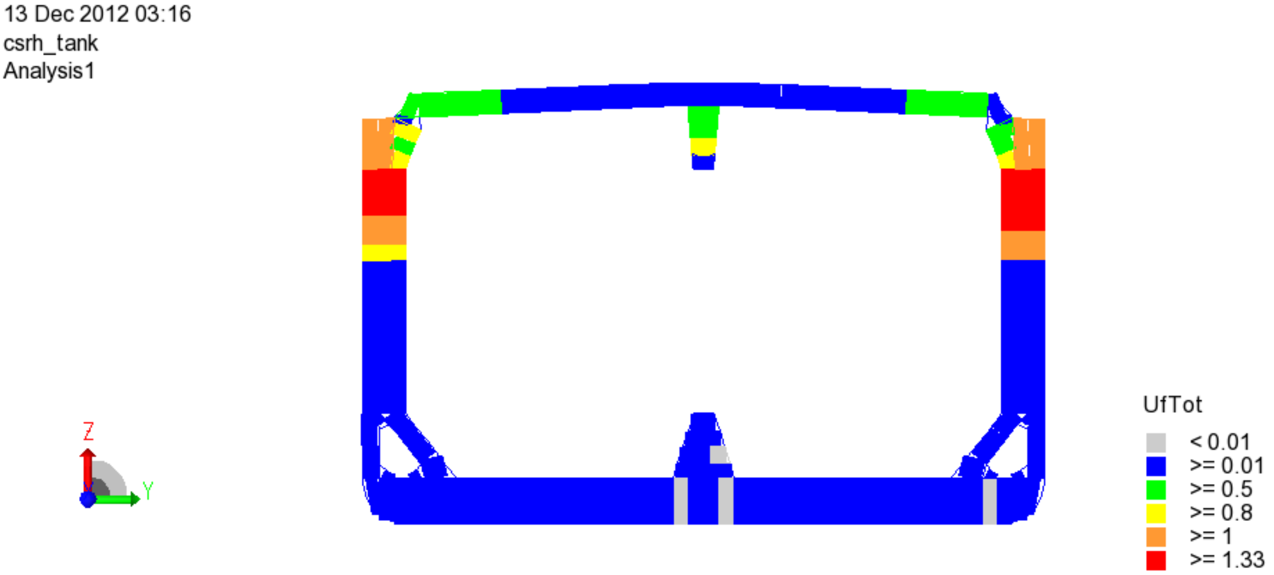




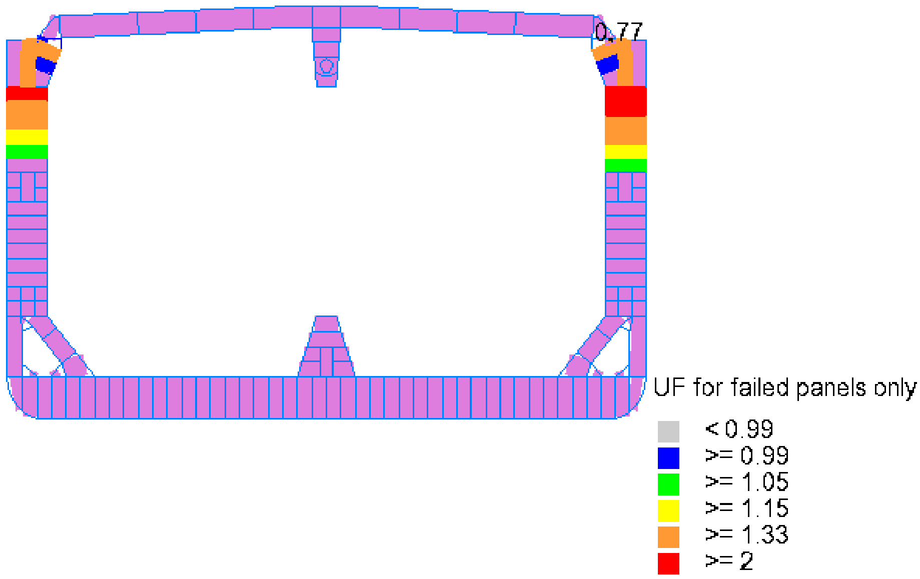
Capacity Model	LoadCase	Position	Status	UfTot	Formula	SubCheck	GeomCheck	Idealization Method
panel(Pl2)	LC05_Double_Mom	0.50	Failed(uf)	1.14	etaBS/allowBS	Plate Buckling		CSRHarmonised Default
panel(Pl3)	LC05_Double_Mom	0.50	Failed(uf)	1.14	etaBS/allowBS	Plate Buckling		CSRHarmonised Default
panel(Pl4)	LC05_Double_Mom	0.50	OK	0.97	etaBS/allowBS	Plate Buckling		CSRHarmonised Default
panel(Pl5)	LC05_Double_Mom	0.50	OK	0.75	etaBS/allowBS	Plate Buckling		CSRHarmonised Default
panel(Pl6)	LC05_Double_Mom	0.50	OK	0.82	etaBS/allowBS	Plate Buckling		CSRHarmonised Default
panel(Pl7)	LC05_Double_Mom	0.50	OK	0.38	etaBS/allowBS	Plate Buckling		CSRHarmonised Default
panel(Pl27, 1)	LC05_Double_Mom	0.50	Failed(uf)	1.14	etaBS/allowBS	Plate Buckling		CSRHarmonised Default

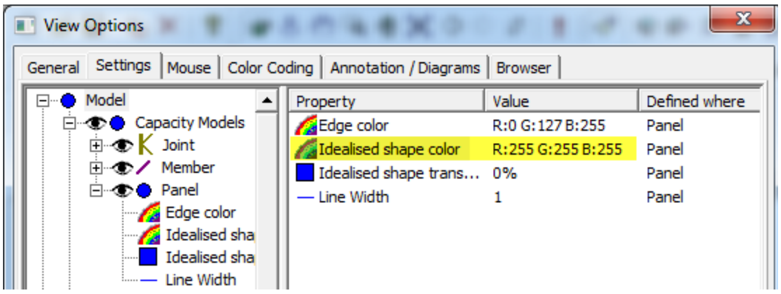


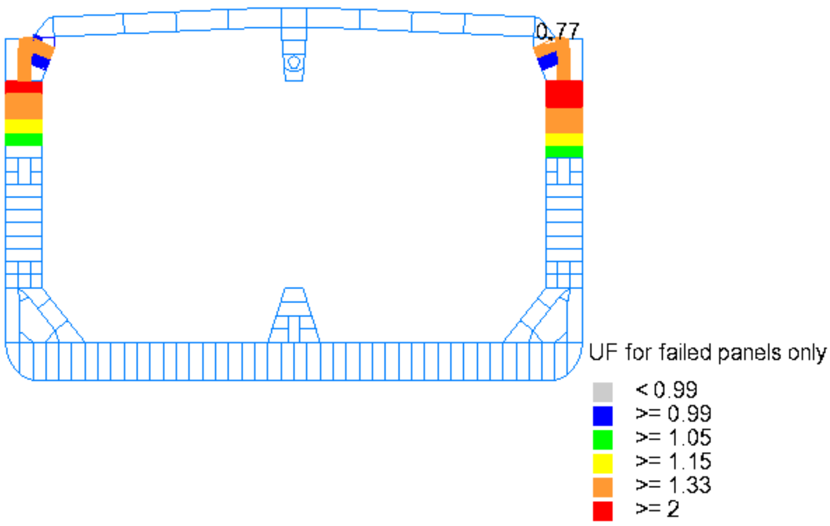
The results can also be colour coded on the capacity model in the graphical window. Select the panels of interest, and click **RMB** in the graphical view to coloucode UfTot



You should repeat the steps in this paragraph for each set of interest,

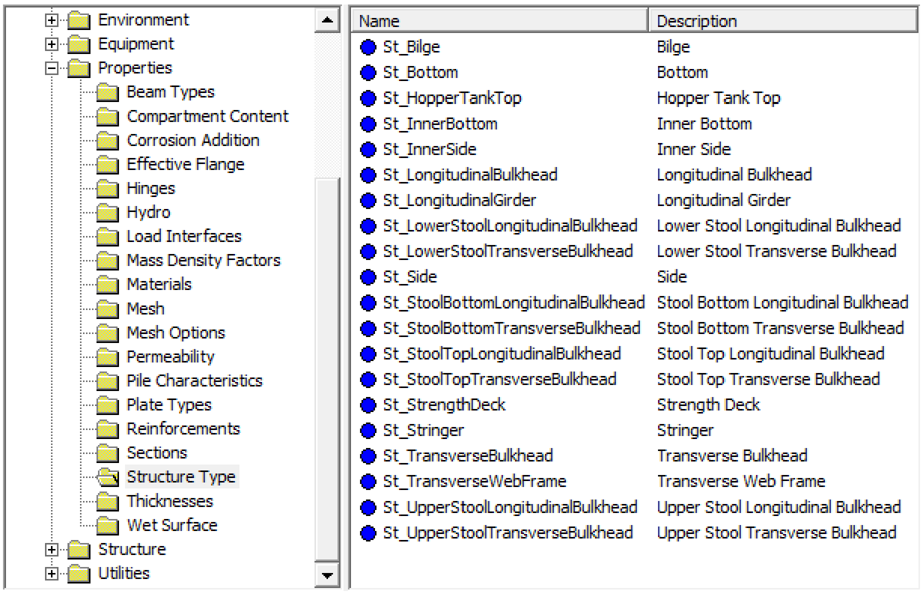

0.77
UF for failed panels only
< 0.99
>= 0.99
>= 1.05
>= 1.15
>= 1.33
>= 2


View Options
General
Settings
Mouse
Color Coding
Annotation / Diagrams
Browser
Model
Capacity Models
Joint
Member
Panel
Edge color
Idealised sha
Idealised sha
Line Width
Property
Value
Defined where
Edge color
R:0 G:127 B:255
Panel
Idealised shape color
R:255 G:255 B:255
Panel
Idealised shape trans...
0%
Panel
Line Width
1
Panel


0.77
UF for failed panels only
< 0.99
>= 0.99
>= 1.05
>= 1.15
>= 1.33
>= 2

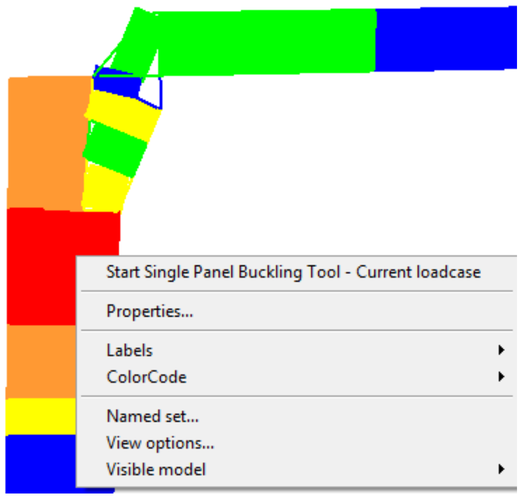
5.6 Structure types and methods

When importing the xml file created by NH, the model will have different structure types corresponding to the different parts of the model. All structure types that are used for your model can be viewed in the browser by clicking *Structure Type* under *Properties*.

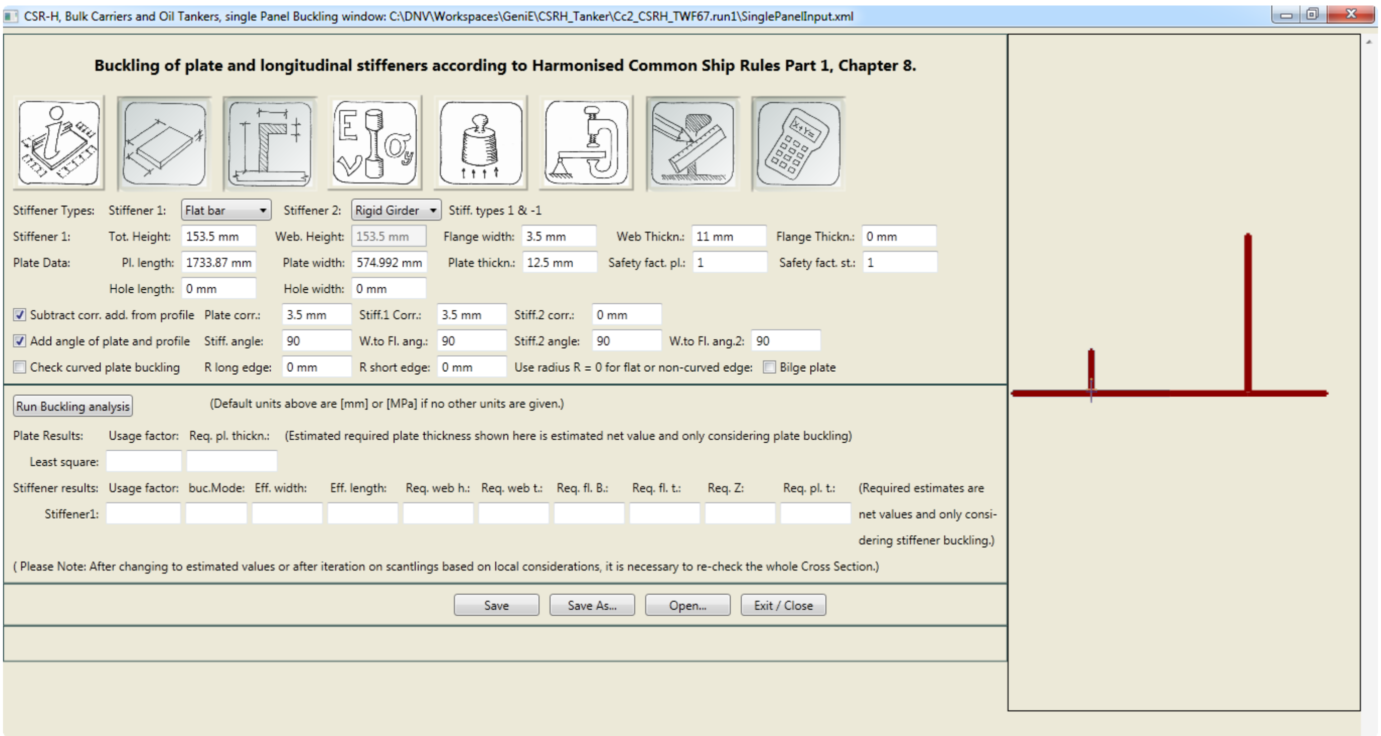


5.7 Starting single panel tool from GeniE

You can start *Single Panel Buckling Tool* from inside GeniE. This can be done either from the graphical window or from the browser. Note that this tool is only available as part of an installation of Nauticus Hull July 2013.

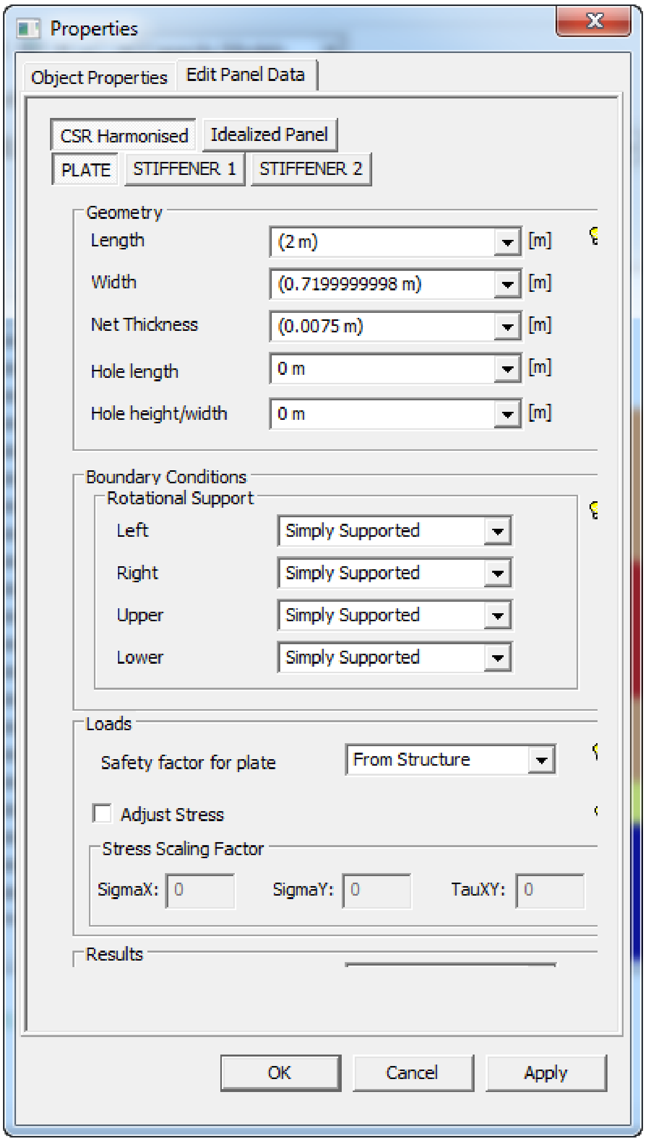


In the graphical window you select a panel, click **RMB** and make your choice. You can choose to open the panel only for the current loadcase



5.8 Changing properties for a panel

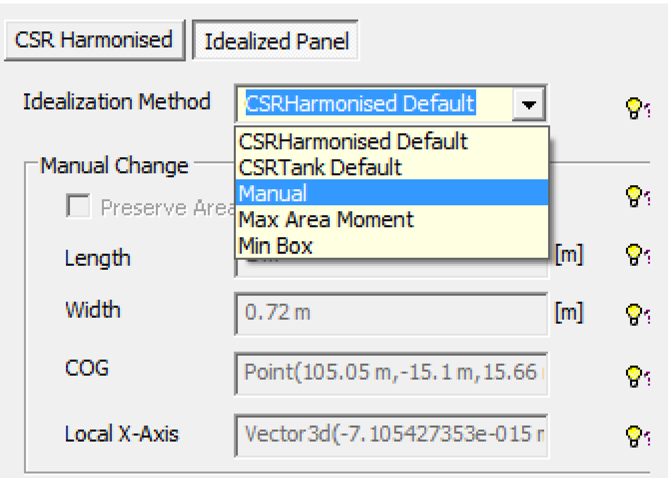
By selecting one or several panels, clicking **RMB** and selecting *Properties*, you get to see the properties of the panel(s) you have selected.



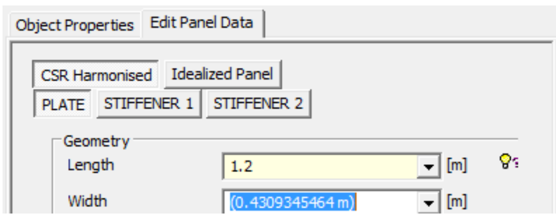
In the example illustration, you can see the properties for one panel.

The values for Length, Width and Net Thickness are shown in brackets. This means that these values are based on the idealized panel.

By clicking on "Idealized Panel" you get to change some of the values for the Idealized Panel. The changes you do will only affect the selected panel(s).



You are able to select idealization method from the available panel idealization methods, CSR BC & OT, (CSR Tank, Max Area Moment and Min Box). In addition you are able to manually set the properties of the panel(s). In this example we are manually changing the length of one panel.

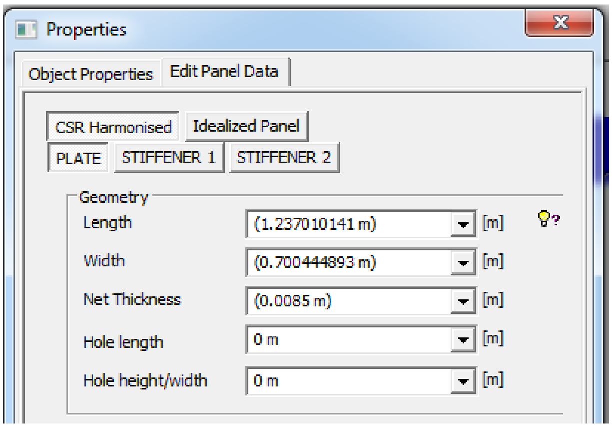


The changes you make here update the Capacity manager and are valid for all Runs.

Changes will be visible in the graphical display.

It is also possible to change the properties for the selected panel(s) directly without changing the idealized panel settings. These changes will only have effect for the current Run.

Geometry -Length/Witdh



From Idealized Shape – Uses the dimensions taken from the properties of the idealized panel. The numbers are shown with brackets around them to indicate they are the same as for the idealized panel.

Manual – You can set your own value manually.

Geometry -Net Thickness

Average from Structure – The net thickness is calculated from the thickness(es) of the conceptual plates from which the panel(s) are derived.

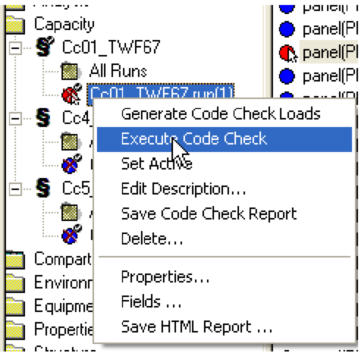
Manual – You can set your own net thickness value manually.

The rest of the properties can be changed in a similar way.

Note that the panels which properties have been changed will not be updated until you re-execute the codecheck.

Capacity Model	LoadCase	Type and Method	Safety Factor	Status	UfTot	SubCheck	NetThick [m]
panel(Pl86)	LC05_Double_Mom	SP - B	1.0	OK	0.85	Plate Buckling	0.0085
panel(Pl217, Pl213)	LC05_Double_Mom	SP - A	1.0	OK	0.78	Plate Buckling	0.0127445
panel(Pl218, Pl219)	LC05_Double_Mom	SP - A	1.0	OK	0.31	Plate Buckling	0.015
panel(Pl223, Pl220)	LC05_Double_Mom	SP - A	1.0	OK	0.74	Plate Buckling	0.0127445
panel(Pl1970, 1)	LC05_Double_Mom	SP - B	1.0	OK	0.03	Plate Buckling	0.0095

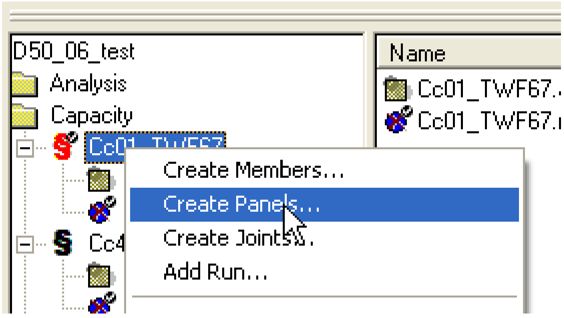
To see any results for the panels you need to select *Execute Code Check* for the appropriate run.



Capacity Model	LoadCase	Type and Method	Safety Factor	Status	UfTot	SubCheck	NetThick [m]
panel(Pl86)	LC05_Double_Mom	SP - B	1.0	OK	0.85	Plate Buckling	0.0085
panel(Pl217, Pl213)	LC05_Double_Mom	SP - A	1.0	OK	0.78	Plate Buckling	0.0127445
panel(Pl218, Pl219)	LC05_Double_Mom	SP - A	1.0	OK	0.37	Plate Buckling	0.015
panel(Pl223, Pl220)	LC05_Double_Mom	SP - A	1.0	OK	0.74	Plate Buckling	0.0127445

Important

The purpose of making changes to one or several panels as described in this paragraph is to test how changing the different properties will affect the usage factor etc for the panel(s). To permanently change the properties for the panels, you need to make changes to the Concept model.



Note that if you select *Create Panels* for your run, all changes you have done to the panels will disappear.

5.9 Investigate and report the results

Please refer to chapter 3.9 for general procedures to investigate results. Here, only the investigation of yield stress analysis results shall be discussed.

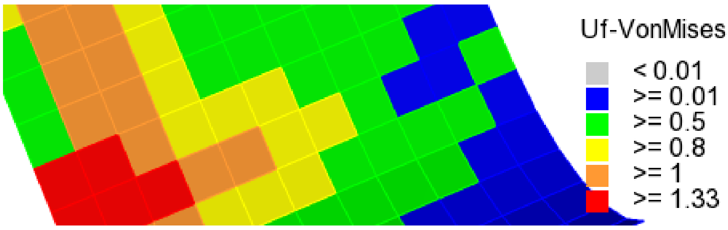
The resulting fraction of von-Mises usage on the permissible usage is stored for each element in the panel and as the worst number for the whole panel. (The worst number, UfTot, is also including the worst buckling utilization if doing buckling check and this is largest.)

The following labels and color code options are per element in the panel:

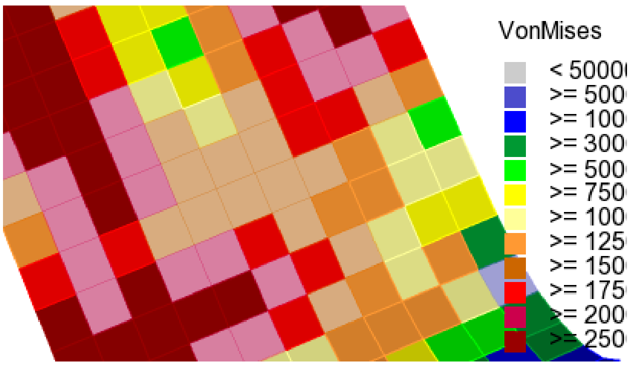
- Uf-VonMises : Von-Mises/(S*YieldStrength) where S is permissible usage for this load case and location.
- VonMises : Average membrane VonMises stress for element.
- Thickness-VonMises : Net element thickness.

Example of color code plots with element borders on the panels:

With the option: “Results - Uf-VonMises”:

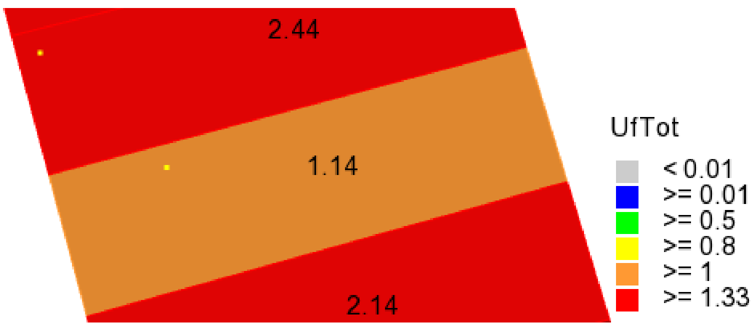


With the option: “Results - VonMises”:



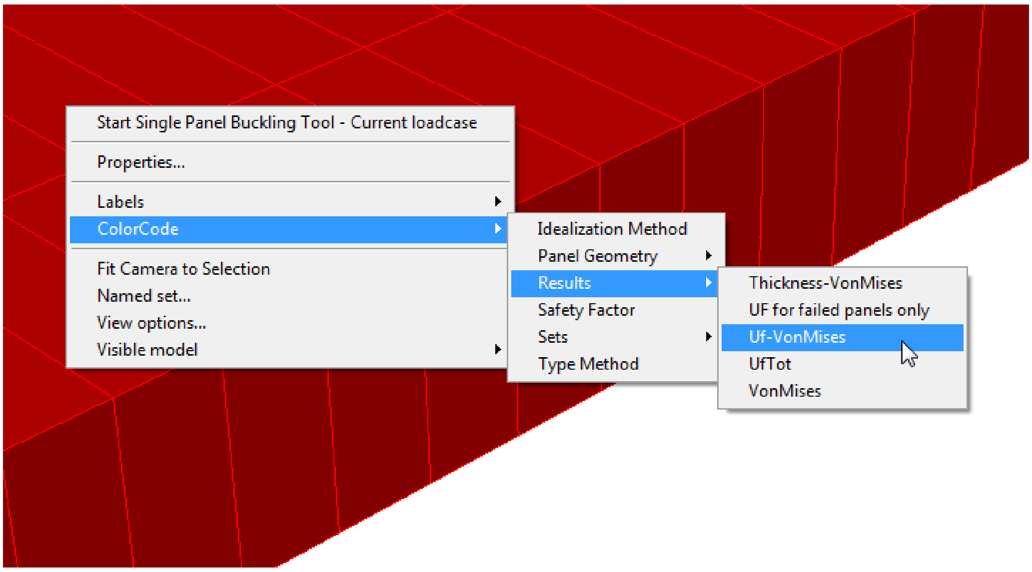
Any other color code or label plot option will be per panel.

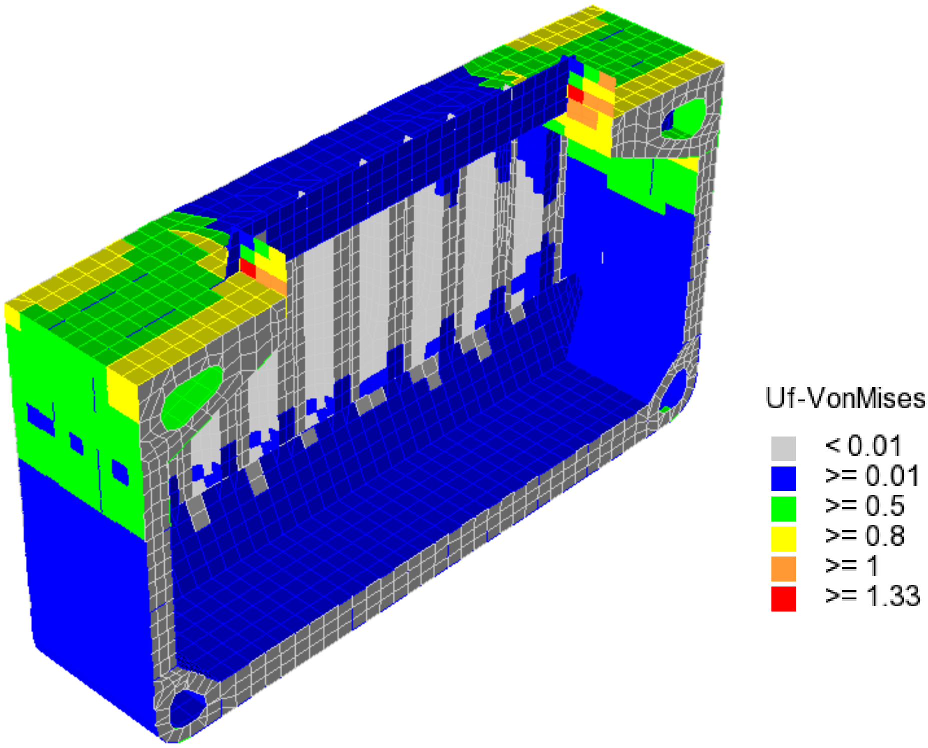
Example: UfTot will show maximum of buckling and yield usage (divided with the relevant permissible usage). Color code will be filling the simplified rectangular panel shape. A small yellow dot of worst position will be shown together with the label when yield stress usage is governing.



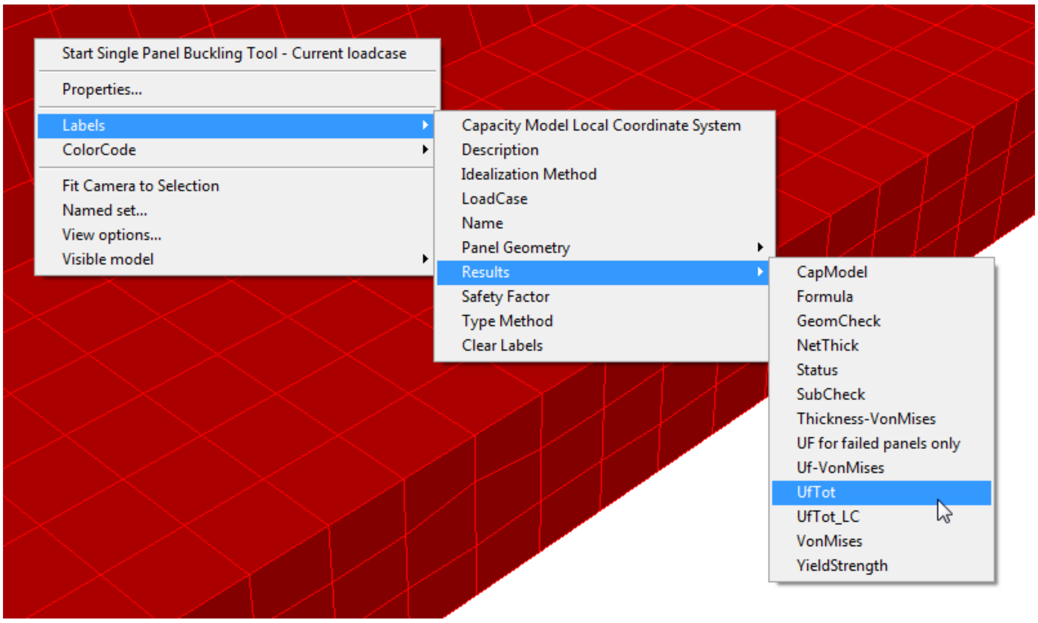
The following example shows a combination of color coding and labelling, combining information about panels and individual finite elements.

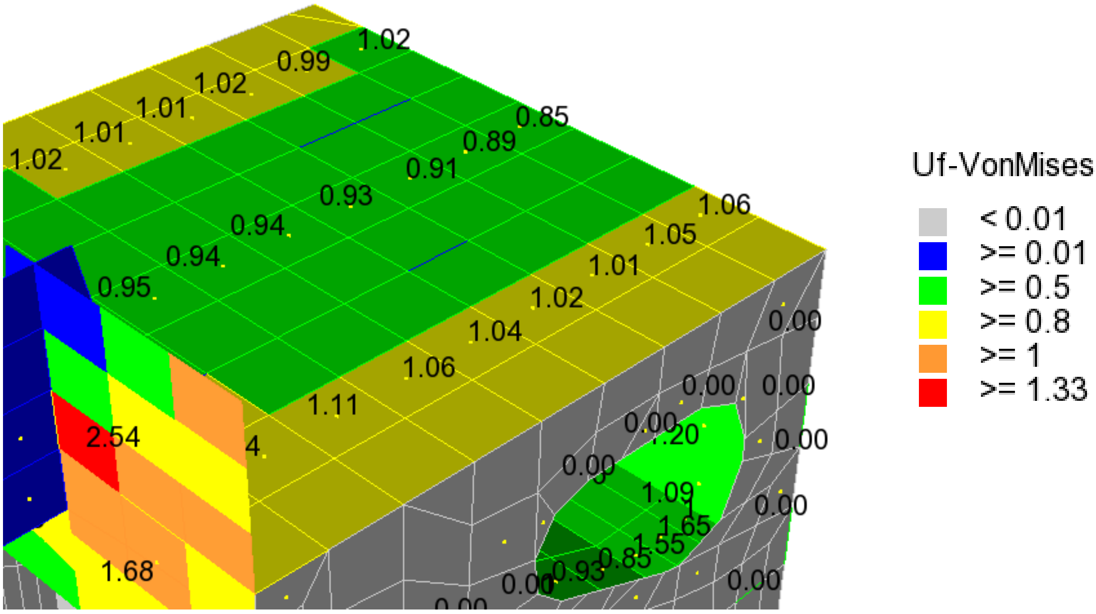
The user can visualize the stress results either directly by Von Mises stress contours (Results – VonMises) or in terms of utilization factor (Results – Uf-VonMises). Both options will be displayed per finite element.



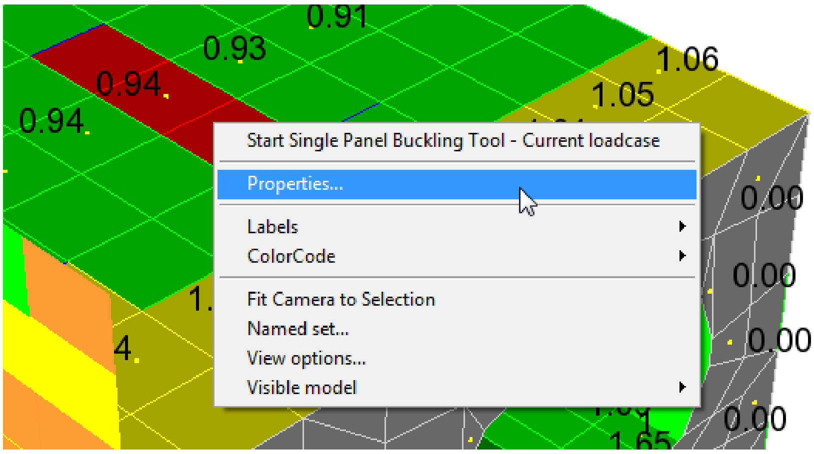


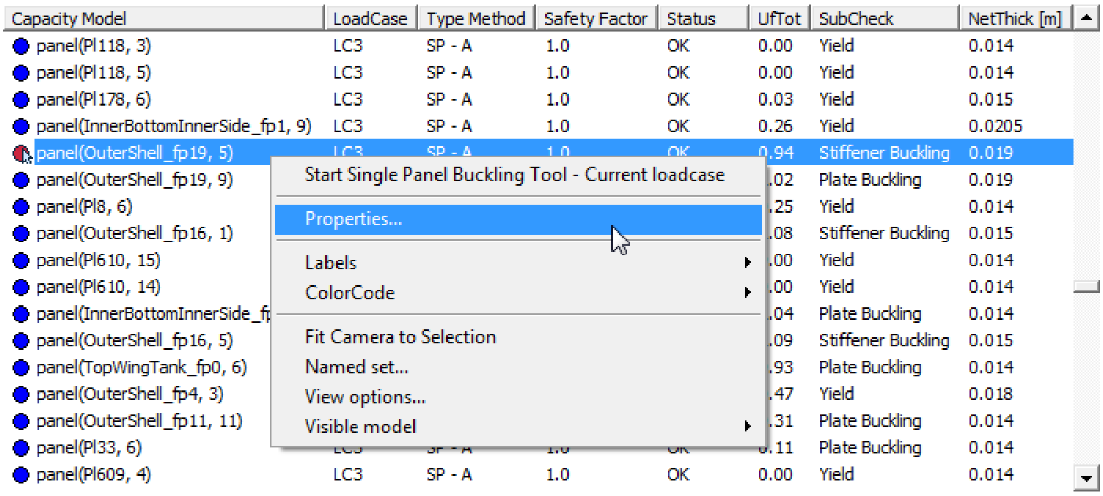
Labels can be added showing the explicit values for various measures. For example, the total utilization factor together with its most critical position for each panel (represented by the yellow point) can be displayed:

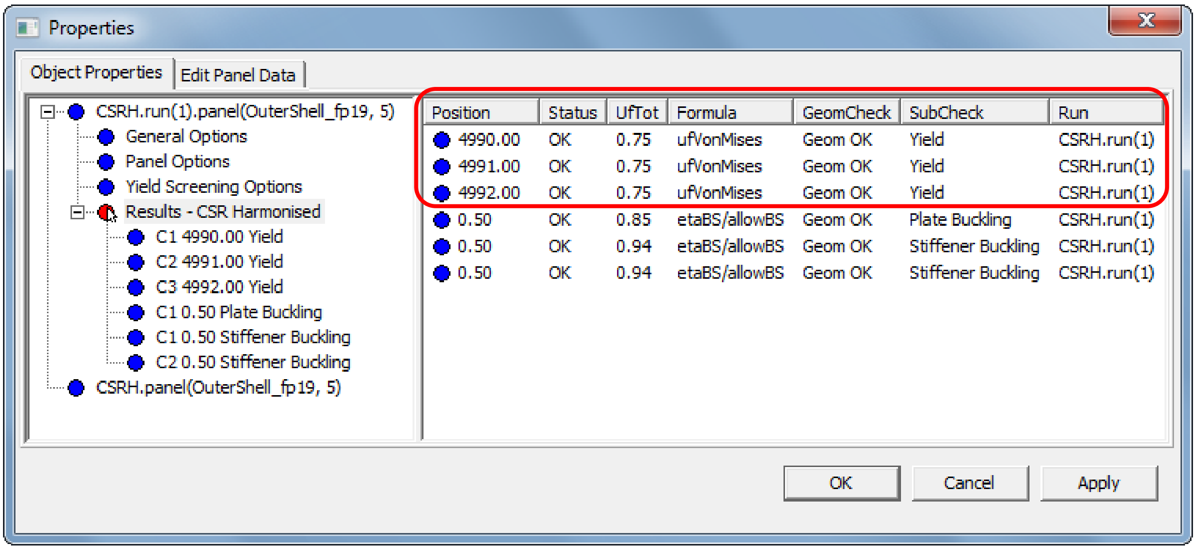




A more detailed description of the code check results can be obtained by entering a panel's properties. One or multiple panels at once can be selected either from the graphics window or the browser. The panel properties can be accessed via a right click on the selection and choosing *Properties* from the context menu. The user can then switch to the *Object Properties* tab to get a detailed list of code check results of the selected panel(s):







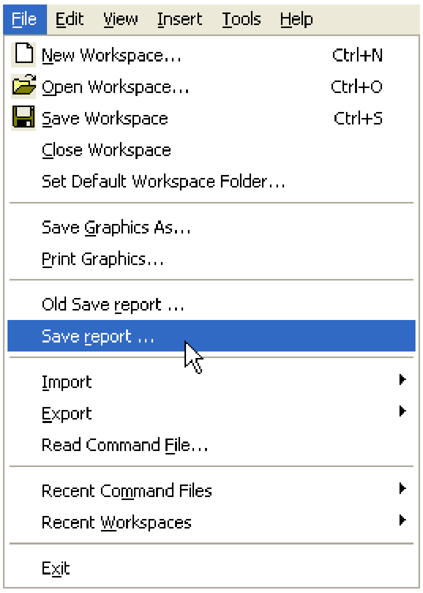
The values in the *Position* column represent the element numbers of all finite elements owned by the selected panel. For those rows showing buckling code check results, this column holds the value 0.5, referring to the middle of the panel. In this view, it can easily be seen which sub check computes the most critical utilization factor.

Make a report CSR BC & OT

A customised report may be generated from the ***File|Save report***.

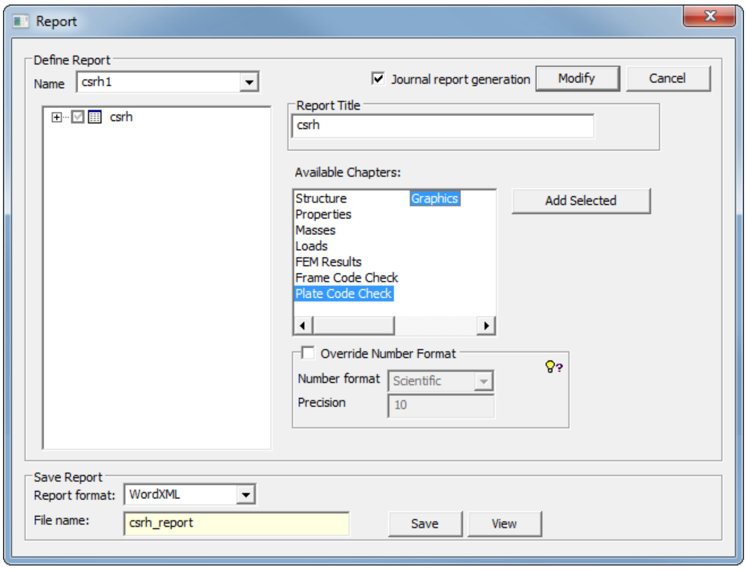
The **File|Save report** allows you to specify the content of your report as well as to decide the levels of detail in of your report.

Here is an example on how to make a code checking report.



To make a report of the plate code check including graphics, you select *Plate Code Check* and *Graphics* under *Available Chapters* and click *Add Selected*.

The report name, title and file name can be changed. By default the file name is the same as the report name.

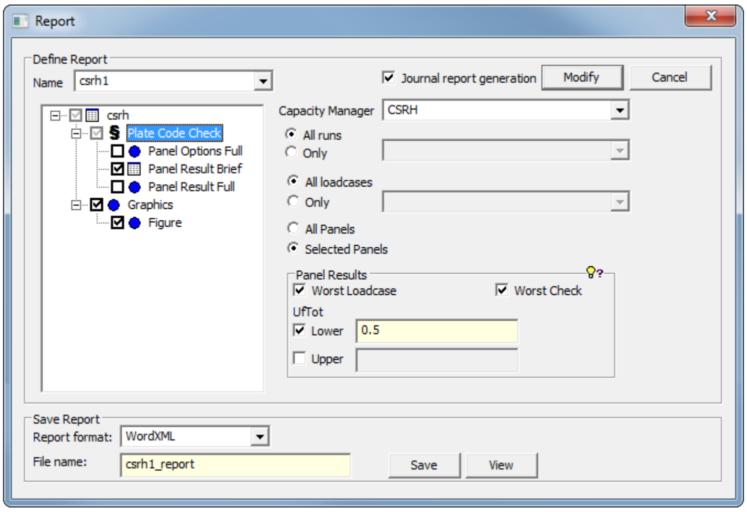


The available tables for *Plate Code Check* are *Panel Options Full*, *Panel Result Brief* (selected in this example) and *Panel Result Full*.

We have checked the *Lower* checkbox for Uftot, and typed in the value 0.5. This means that the report will not list any panels having a Uftot value of 0.5 or lower.

Note that the number of panels included in the report can be further limited by selecting only the panels of interest before entering the report dialog and then choose *Selected Panels*.

We have chosen the report format to be *WordXML*. Other available formats are: *CSV*, *HTML*, *ExcelXML* and *Text*.



In the *Graphics* settings the user can select which load cases shall be considered for the picture creation.

Figure settings give control over the color coding and labelling options all details regarding the setup of figures to be included in the report.

After having clicked *Save*, you can have a look at the report by clicking *View*.

Below is an example of what an excerpt of our report might look like in the report. For the graphics, a customized camera position was chosen by using the *Snapshot* option in the *Figure* settings.

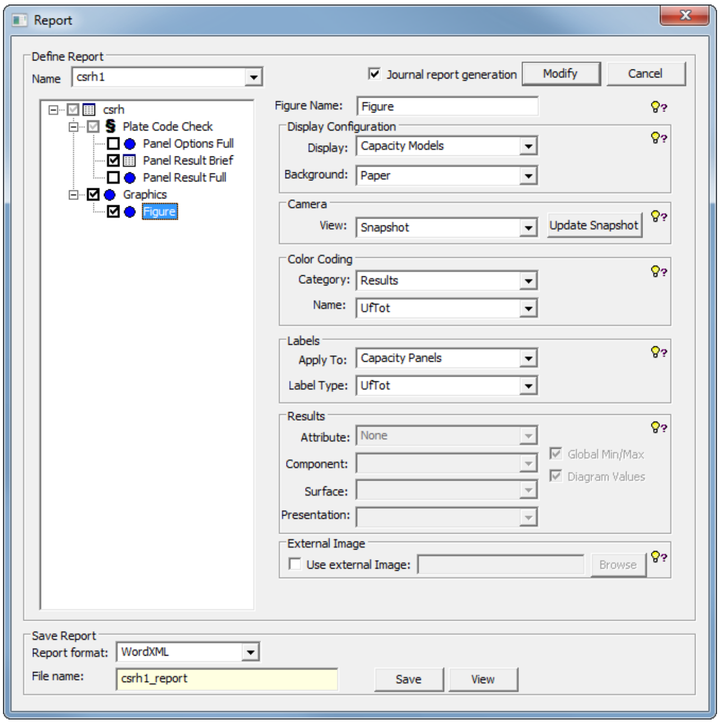


Table of Contents

1 CSRH : Plate Code Check

Description : Capacity Manager

1.1 CSRH.run(1) : Plate Code Check

Description : CSR Harmonised

1.1.1 CSRH.run(1) : Panel Result Brief

CSRH.run(1) : Panel Result Brief

- Sorted by UfTot (Descending)
- Then sorted by LoadCase (Ascending)
- Filtered by Limit : (UfTot >= 0.5)
- Run : CSRH.run(1)
- Worst LoadCase per Panel
- All SubChecks per Panel
- Worst of Whole Panel and SubPanel Checks per Panel

Panel	LoadCase	Position	Status	UfTot	Formula	GeomCheck	SubCheck	Run
Pl934	LC3	0.50	Failed(uf)	2.38	etaBS/allowBS	Geom OK	Stiffener Buckling	CSRH.run(1)
Pl934	LC3	0.50	Failed(uf)	2.37	etaBS/allowBS	Geom OK	Plate Buckling	CSRH.run(1)
Pl947	LC3	0.50	Failed(uf)	1.52	etaBS/allowBS	Geom OK	Plate Buckling	CSRH.run(1)

2.3 Figure_LC3

23 Jan 2015 10:22
csrh
Analysis1
LC3
FEM Loadcase = 3
Force: [N], Length: [m]

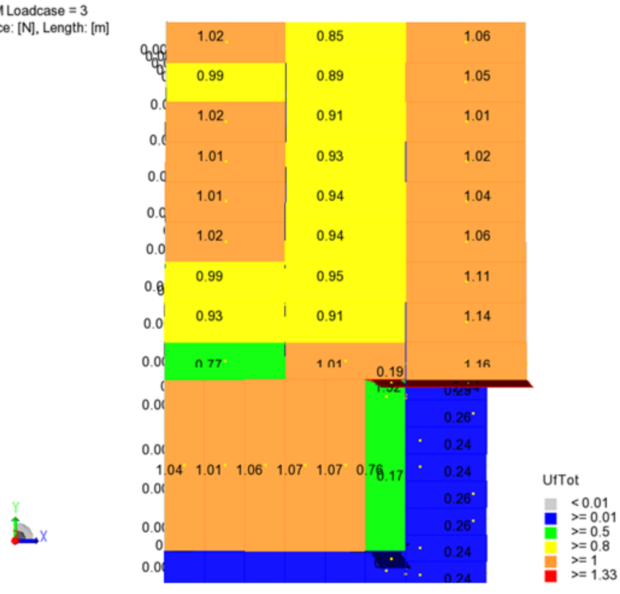


Figure (Loadcase: LC3)

CSR BC & OT code check results report

Detailed results are shown in 3 tables. These are one for yield screening assessment, one for plate buckling and one for stiffener buckling.

Ref to rules

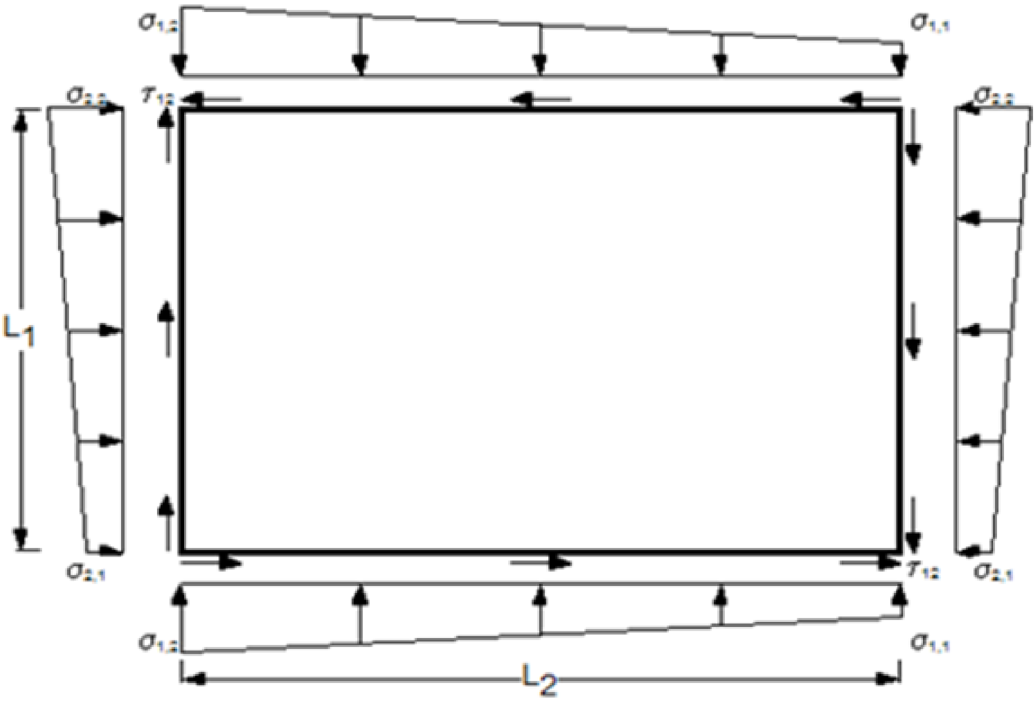
Panel	The name of the panel
Loadcase	The name of the loadcase
Position	Centroid of panel. 0.50 is on the middle of panel. For yield assessment: element id.
Status	Status of a panel is either: *OK*, *Failed (Uf)* or *Failed (geo)*
UfTot	Utilisation factor, worst of yield, plate and stiffener buckling in summary table.
Formula	The formula that is governing
GeomCheck	Shows whether the panel’s geometry is consistent with regards to the code check
SubCheck	Which sub-check that is governing. (Not relevant for CSR-BC & OT.)
Run	The name of the run
Governing Method	Assessment Method A or Method B. Ref rules Pt.1 Ch.8 Sec.1 [3.1.3]
etaBS/allowBS	Utilisation BS (Buckling Strength)
ufVonMises	Utilisation for Yield assessment
Idealization Method	One of the methods available during panel creation. (CSRBC & OT Default, Irregular or One panel per plate).
Panel Length	Length of panel. Should be larger than Panel Width.
Panel Width	Width of panel.
Panel Thickness	Plate thickness used in buckling calc. (net, full corrosion addition is subtracted)
Thickness (-tc/2)	Plate thickness used in stress assessment. (Half of corrosion addition is subtracted).
Stiffener Type	F=Flatbar, L=Angle, T=Tee
Stiffener Boundary	S=Sniped, C=Continous, F=Fixed
Stiffener Height	Stiffener Height
Web Thickness	Web Thickness (net)
Flange Width	Flange Width
Flange Thickness	Flange Thickness (net)
E-Modulus	
Poisson’s Ratio	
Yield Stress Plate	
Yield Stress Stiffener	

Support Left	Rotational support of plate edge.
Support Right	(Simply supported, Fixed or Free)
Support Upper	
Support Lower	
No. of Tripping brackets	Number of tripping brackets. Relevant for transversely stiffened single sides in bulk skips only.
Sigma11	Axial stress - Stress in X direction along short side, *see illustration*
Sigma12	Axial stress - Stress in X direction along short side, *see illustration*
Sigma21	Transverse stress - Stress in Y direction along long side, *see illustration*
Sigma22	Transverse stress - Stress in Y direction along long side, *see illustration*
Tau12	Shear stress in panel
Pressure	Lateral pressure on panel
vonMises	Actual von Mises stress.
etaBS	Actual usage factor for Buckling Strength calculations.
allowBS	Allowable usage factor for Buckling Strength calculations.
allowUF	Allowable usage factor in Yield stress assessment.
Required Thickn.	Roughly estimated panel thickness required to get usage factor to be 1.0
Required Web Height	When given, roughly estimated required web height of stiffener required to get stiffener buckling usage factor to be 1.0
Required Web Thickn.	When given, roughly estimated required web thickness of stiffener required to get stiffener buckling usage factor to be 1.0
Required Flange W.	When given, roughly estimated required Flange width of stiffener required to get stiffener buckling usage factor to be 1.0
Required Flange T.	When given, roughly estimated required Flange thickness of stiffener required to get stiffener buckling usage factor to be 1.0
Required Plate T.	When given, roughly estimated required plate panel thickness required to get stiffener buckling usage factor to be 1.0
Required Z (estimate)	When given, roughly estimated required Sectional moment of stiffener required to get stiffener buckling usage factor to be 1.0
Actual Z (net)	Actual sectional moment of stiffener with corrosion subtracted.

If selecting the report option – Plate Code Check, with sub-options Panel Result Brief or Panel Result Full, then only the worst yield result per panel will be reported (but only if yield check is governing or you perform a Yield check only). The FEM element id will be shown in the Position column."

On the General Tab of the Code Check, there is an option to execute Yield Check only. This option is convenient to use when you only want to assess the Yield Check results in detail.

Unstiffened panel:



The left, right, upper and lower input of the boundary conditions corresponds to the sides of these illustrations.

5.10 Yield screening

Yield screening: Ref rules: Part 1, Chapter 7, Section 2.

This yield check is currently implemented for elements of panels (and optionally beams on boundary of panels).

4 types of screening:

- Manual check
- Coarse mesh yield check.
- Yield Screening (not applicable for DNV GL rules).
- Fine mesh yield check.

All checks are based on FEM analysis results:

- For plate elements, the Membrane Stress at element centroid is used. Plate thickness and material yield strength is picked from the FEM model properties.
- For beams, the computed Axial stress (assume no bending) is used for the yield check. Section area and material yield strength is picked from the FEM model properties.

5.10.1 Manual check:

User specifies permissible usage factor for Seagoing and Harbour loads.

5.10.2 Coarse mesh check:

Ref: Part 1, Chapter 7, Section 2, §5.2. Yield strength assessment and acceptance criteria for a coarse mesh size of approximately (stiffener spacing) x (stiffener spacing). (Or averaged if smaller elements, but this might be difficult to achieve. At least skipped in first version.). Required input, default values and how they might be found automatically:

a. isCorrugationWithLiquidLoads(false), Might be based on general option “corrugated bhd” option, (alternatively a new structure type), and checking if element has pressure, (Alternatively checking if there are compartments on each side of element that has liquid loads). Is it ok to use different permissible UF above and below liquid surface in partly loaded compartment?
b. hasLowerStool(false), Might scan whole model to see if the structure types: “Stool Bottom Transverse Bulkhead” or “Stool Bottom Longitudinal Bulkhead” has been used. Can a ship have different Corrugated bulkhead types, so we need to check x and y coordinates also?
c. isLowEndSupportsForCorrugation(false), More difficult, but might check if element is within half web frame or 3 stiffener spacing from any structure that has a new structure type for corrugated bulkhead.
d. isDummyRodOfCorrugatedBhd(false), Might be used for all stiffeners on corrugated bulkheads..
e. hasUpperAndLowerSlotsForStiffeners(false), Difficult, may require new structure types, other options or user input. Might assume true for structure type “Transverse wash bulkhead” and false for everything else…?

f. hasOpenSlots(false), Same as above.

g. hasOpening(false), May use new opening concept or reuse existing code in CSR Bulk that looks for internal loops in panel geometry.

h. opening_h(0.0), Now setting this to panel width

i. opening_h0(0.0), Now Copying input to buckling. Might use data from new opening concept or try to calculate from geometry inner loop.

j. opening_l0(0.0), Same as above.

k. opening_A_shr_n50(0.0), (or h_eff ?). This is user input. Must be taken from drawings and based on simplification when modelling.

l. opening_A_FEM_n50(0.0), // Now calculated as: h * t_mod_n50 ?

m. t_mod_n50(0.0), Now found from element thickness.

5.10.3 Yield screening

(This is screening done on a coarse model. Used to find out where fine mesh modeling is required.) Ref: Part 1, Chapter 7, Section 3. Required input, default values and how they might be found automatically:

a. isOutsideMidshipCargoHoldRegion(false),
b. detailType(OpeningInPrimarySupportMemberWeb), Difficult, might need new structure types or other position properties to cover all options:

// Within cargo hold area:

OpeningInPrimarySupportMemberWeb,

OpeningInMainBracketAndButtresWeb,

BracketToe,

HeelOfTrvBhdHorStringer,

HeelOfTrvBhdHorStringerAtLongBhdHorStringer,

ConnectionAtStoolOrCorrugation,

ConnectionAtTopsideTank,

HatchCornerArea,

// Outside Midship Cargo Hold Region

HopperKnuckle,

SideFrameEndBracket,

LargeOpenings,

CorrugatedBhdConnection,

HighCoarseMeshYieldAreaInMidshipregion

c. coarseMeshYieldUtilisation(…) ? User input
d. coarseMeshPermissibleScreeningFactor, see above for required input.
e. sigmaFM_fineMeshVonMisesInMidshipRegion, ? user input ?
f. sigmaCM_coarseMeshVonMisesInMidshipRegion, ? user input ?
g. sigmaC_coarseMeshVonMisesAtDetail, ? user input ?
h. isCriticalFatigueDetail, ? user input ?
i. opening_r(0.0), from new opening concept or might try to calculate based on inner loop geometry…?
j. opening_h; See coarse mesh input.
k. opening_h0; See coarse mesh input.
l. opening_l0; See coarse mesh input.
m. opening_A_shr_n50; See coarse mesh input.
n. t_mod_n50; See coarse mesh input.
o. bracketToe_b1, ? user input ? Or might try to calculate based on geometry?
p. bracketToe_b2, ? user input ? Or might try to calculate based on geometry?
q. bracketToe_A_beam_n50(0.0), Might be found from one of the panel stiffeners
r. bracketToe_Ra_in_mm, ? user input ? Or might try to calculate based on geometry?

5.10.4 Fine Mesh check:

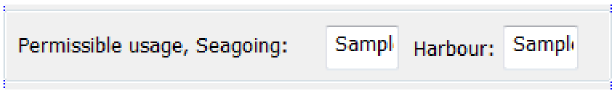
Ref: Part 1, Chapter 7, Section 3, §6. Yield strength acceptance criteria for a fine mesh size of 50 x 50 mm. (Or averaged if smaller elements, but this might be difficult to achieve. At least skipped in first version.). Required input, default values and how they might be found automatically:

a. isAdjacentToWeld(false), might check distance from element center to panel border or plate seam. (Difficult).
b. isCriticalFatigueDetail(false), user input?
c. isAreaWhereLowerStoolIsNotFittedToCorrugatedBhd(false), user input? Might check that element has structure type: “Stool Bottom Transverse Bulkhead” or “Stool Bottom Longitudinal Bulkhead”, and that there are no corrugated bulkheads nearby. Will need new structure type(s) for corrugated bulkhead.

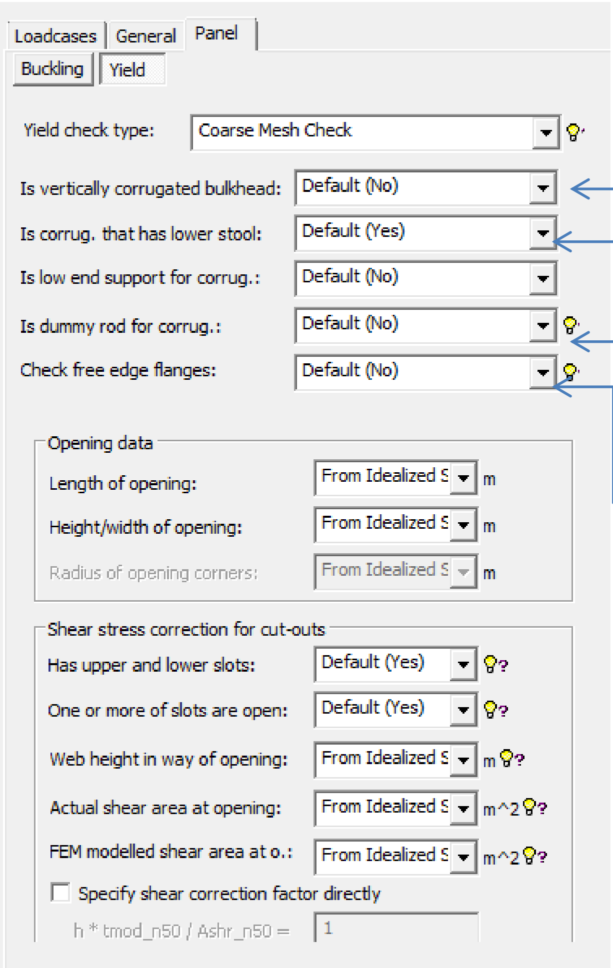
5.10.5 Views:

Selections in this trigger display of one of the four input options below:

1. Manual input:



2. Coarse input:



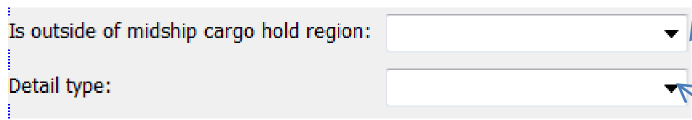
If yes here, then slots, opening data and shear correction inputs are disabled or hidden

If no here, then the rest of the corrugation input fields, (2 next inputs), are disabled or hidden

If yes here, then **all beams on panel boundaries** are checked. The user shall only answer yes for panels with dummy rods.

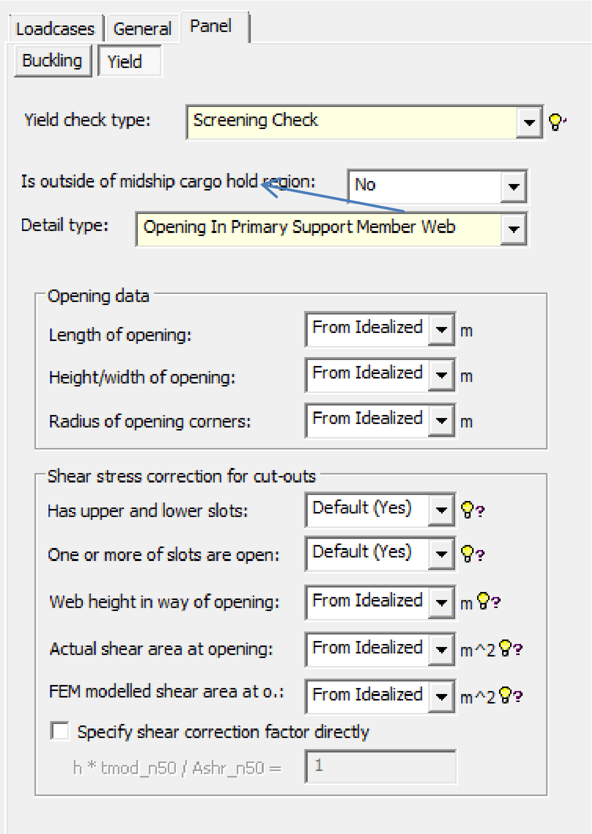
If yes here, then **beams on free panel boundaries** are checked. The user shall only answer yes for panels with flanges to be checked.

3. Screening



Selection here affects number of detail types in next combo:

Selection here decide which of the following parts are displayed or hidden:

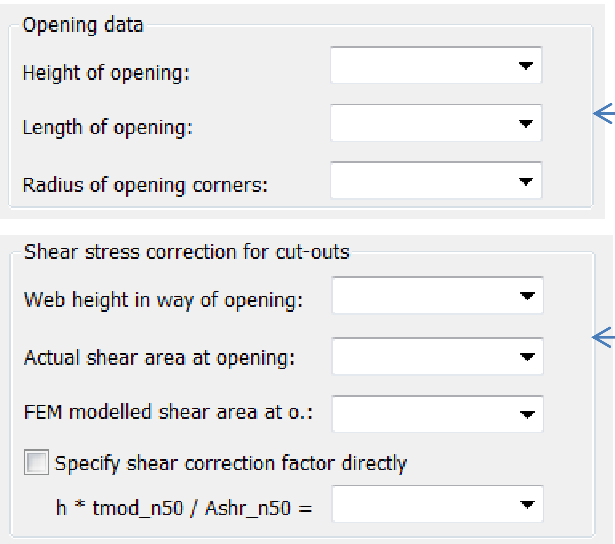


Coarse mesh input required for:

Manholes,
ConnectionAtStoolOrCorrugation,
ConnectionAtTopsideTank,
HatchCornerArea,
HopperKnuckle,
SideFrameEndBracket,

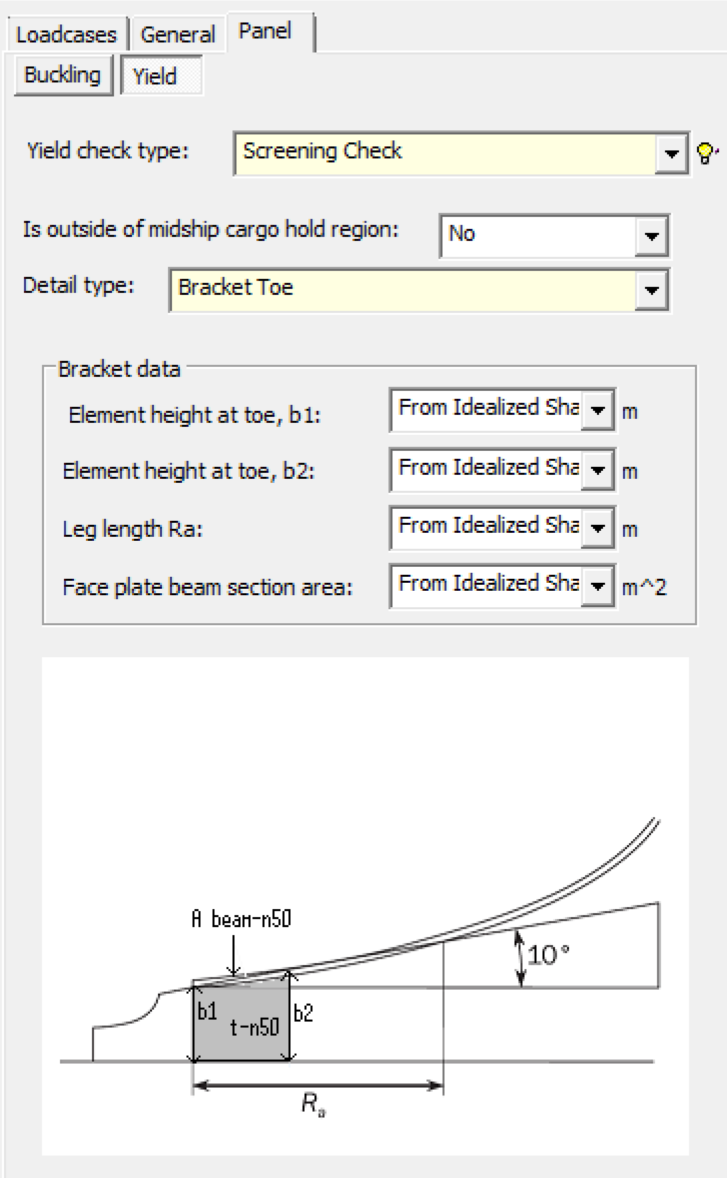
No additional input is required for:

NoScreeningIsRequired,
HeelOfTrvBhdHorStringer,
HeelOfTrvBhdHorStringerAtLongBhdHorStringer,



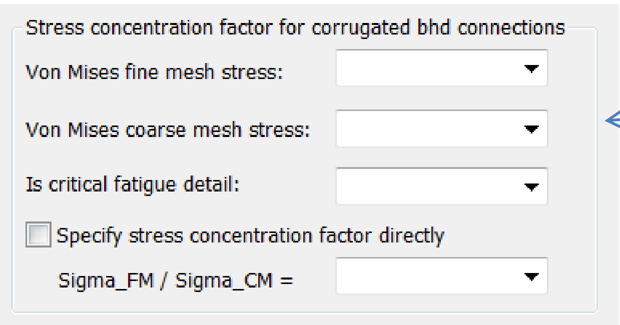
Opening data, including radius and shear correction is required for:

OpeningInPrimarySupportMemberWeb,
OpeningInMainBracketAndButtresWeb,



Bracket toe data is required for:

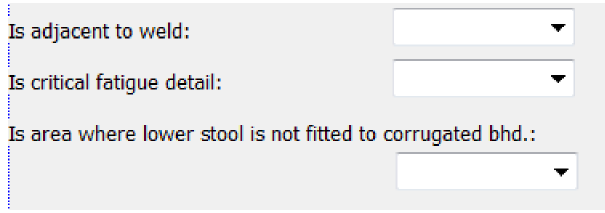
BracketToe,



Stress concentration data is only required for:

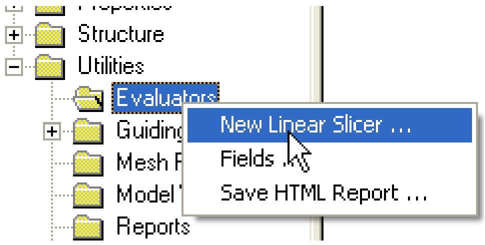
CorrugatedBhdConnection,

4. Fine Mesh:


Is adjacent to weld:
Is critical fatigue detail:
Is area where lower stool is not fitted to corrugated bhd.:

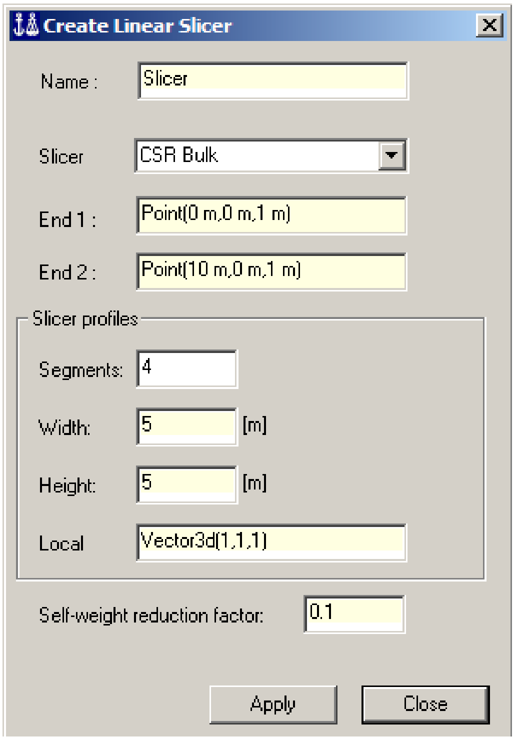
6 OTHER CONCEPTS

6.1 Linear Slicer



The linear slicer is used to calculate the shear force and bending moment distributions at distinct positions along an axis, defined for an elongated structure, which can ideally be regarded as a beam. The linear slicer has been especially designed for Bulk Carriers and Tankers, in accordance with the Common Structural Rules.

Selecting the menu item “Insert\Linear Slicer” or by selecting the tree-node “Utilities\Evaluators” in the GeniE browser tree and the menu item “New Linear Slicer” on the RMB, one can create the slicer, with the aid of the dialog shown in the next figure.

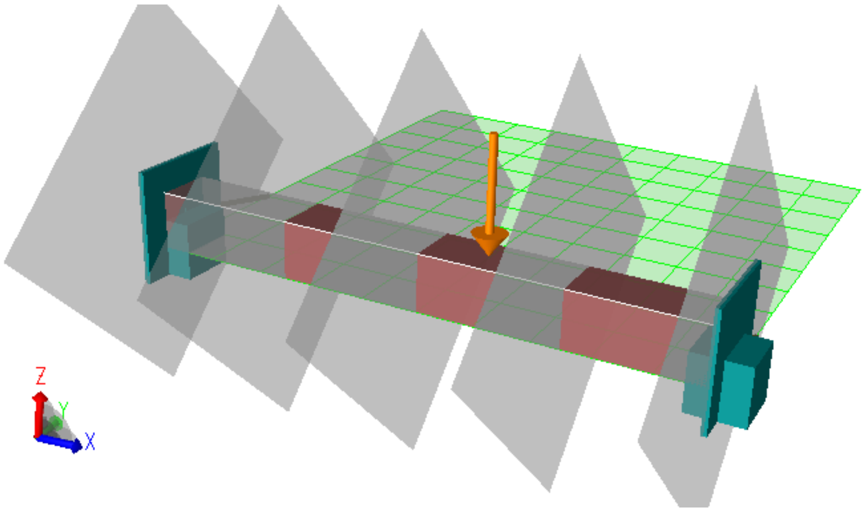


The points *End1* and *End2* define an axis of the structure, along which GeniE subdivides the structure in equidistant *Segments*. The subdivision is done by slicing the structure with planes perpendicular to the axis. These planes are shown graphically as rectangular planar regions, with dimensions of $Width \times Height$. The orientation of these regions is governed by the *Local* vector. The js-code generated for this operation is the following:

```
Slicer = LinearSlicerCSRBulk(Point(0 m,0 m,1 m), Point(10 m,0 m,1 m), Vector3d(1,1,1), 5, 5);
Slicer.segments = 4;
```

```
Slicer.reductionFactorSelfWeight = 0.1;
```

For a simple beam-like structure, the above choices give the result, shown in the following figure.

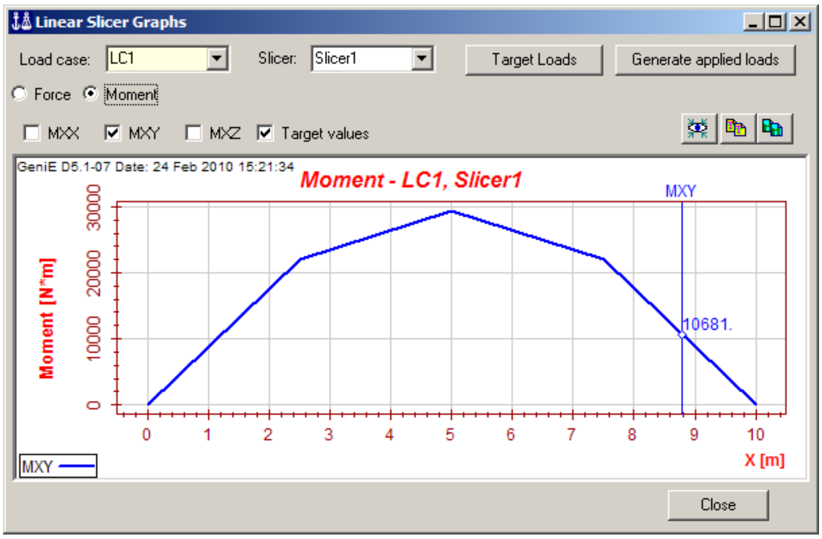


Selecting the slicer in the GeniE browser and pressing the RMB, we are given the following choices:

- "Rename" for changing the slicer name. This can equivalently be done through the scripting engine, by using the command:

  ```
  Rename(Slicer,"Slicer1");
  ```

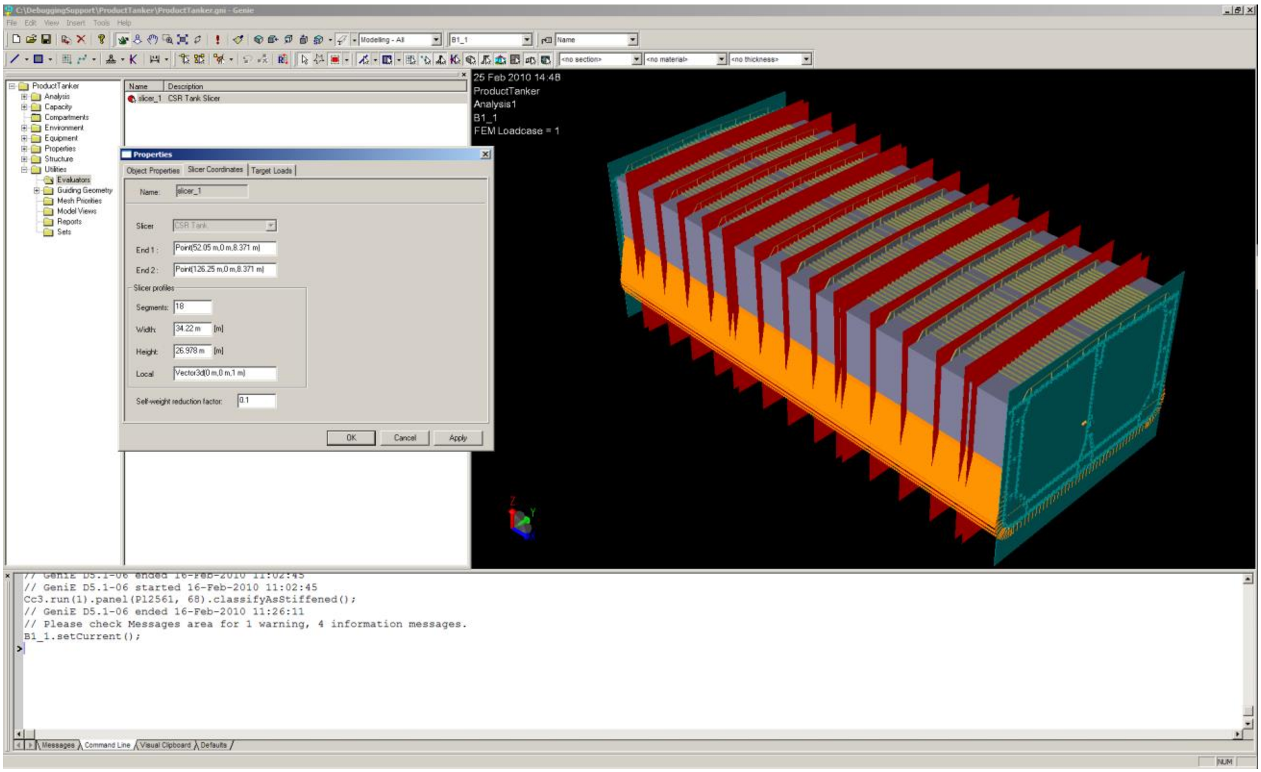
- "Properties", which shows up a dialog for changing the parameters of the linear slicer definition.
- "Slicer Graphs", which shows up a dialog for the graphical representation of the results, i.e. the shear forces and bending moments at the intersections of the planes-slices and the structure axis. The results for the bending moment, for the above example, are shown below.



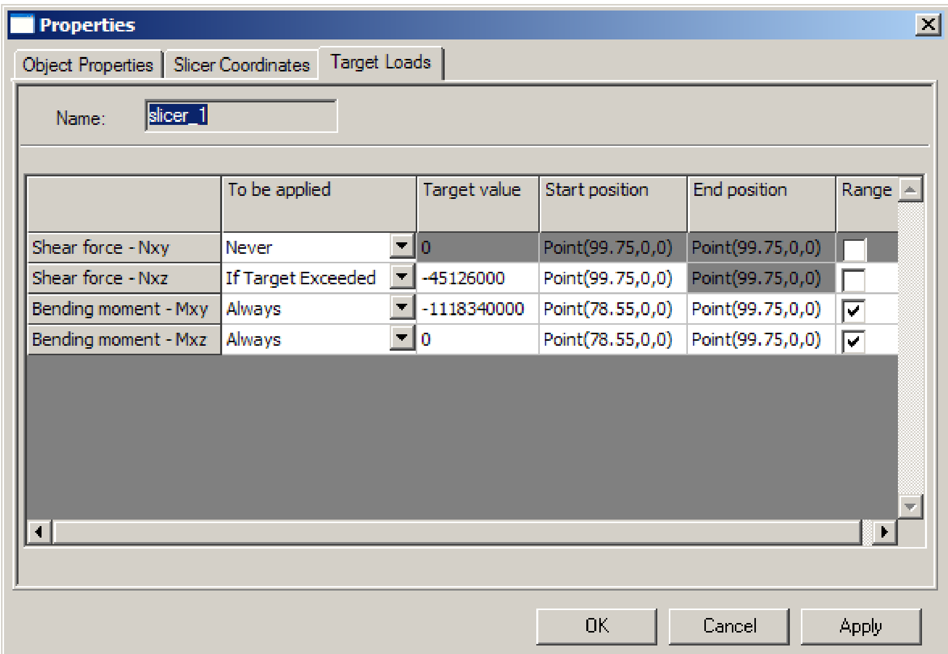
Clicking the "Target Loads" button in the above dialog, the user is able to change the values of shear forces and/or bending moments at specific positions and GeniE calculates the correction moment and shear forces to achieve the given target values. The correction moments are automatically applied to the model.

Application of Target Loads in accordance with the CSR for Tankers over 150 m (Pt.8 Ch.1 App. B)

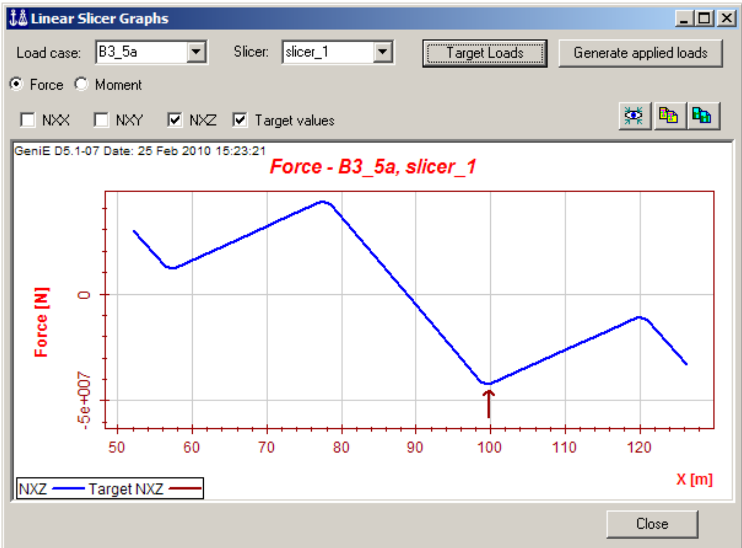
Import the “Rule Loads XML” file, from Nauticus Hull (NH_RuleLoads.xml). In this file there is enough information for the creation of the linear slicer. Moreover, the slices can be defined at non-equidistant positions in Nauticus Hull. The next figure shows three compartments of a Product Tanker amidships. Around and closely to each transverse bulkhead two slices (red planar regions) have been defined, while inside the compartments the slices are equidistantly distributed.

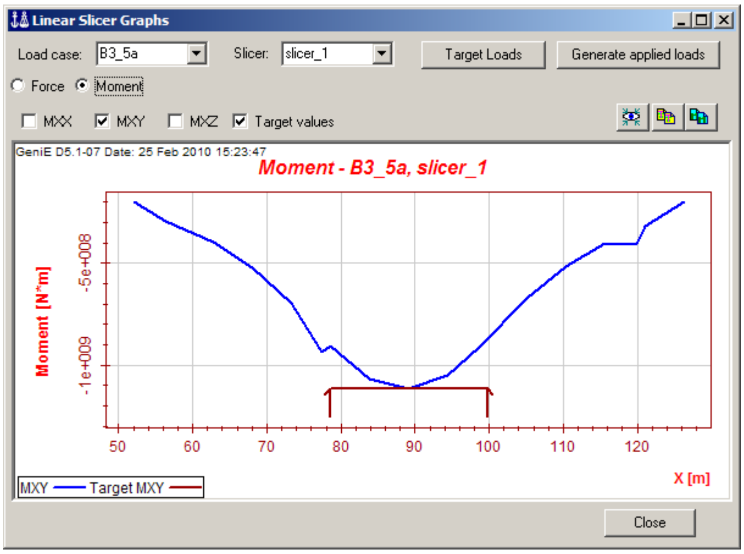


By selecting the slicer and the menu item “Properties” on the RMB or the menu item “Slicer Graphs” on the RMB and then pressing the button “Target Loads”, the following dialog is shown up.



The user can see, check and change -if needed- the values of the shear force (-45126000 [N] at 99.75[m]) and the bending moment (-1118340000[Nm] between 78.55[m] and 99.75[m]) which come directly from Nauticus Hull. Then, GeniE tries to redistribute the shear forces and bending moments so as to achieve these restrictions. The results are shown in the next figures, for the example herein.





6.2 Hull Girder Load Adjuster

For CSR-H there is a separate tool instead of the integrated Linear slicer. This tool is started from the Nauticus Hull FEA-template and is stated between the meshing/load generation and the analysis. It is described in a separate user guide.

From GeniE V7.3 the HGLA may also be executed directly from GeniE. See GeniE User Manual Vol 1 chapter 2.16 - Hull Girder Load Adjuster (HGLA) for further information.

6.3 Co-sentric stiffener

6.3.1 Some definitions

Co-centric stiffener is a technique widely used on the modulation on stiffened plates. This technique simplifies the modeling of stiffened plates without the need of a correct stiffener position to the plate.

Definition: The co-centric beam profile should be represented as an equivalent **general beam** with **area** equal to the area of eccentric beam (**excluding** effective flange from the plate) and bending stiffness = the bending stiffness of the stiffener (**including** effective flange from the plate).

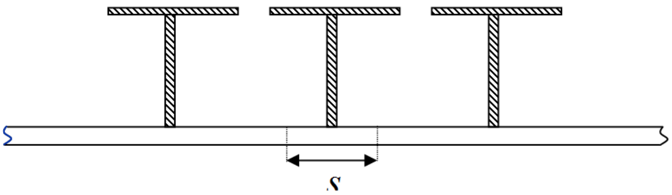


Figure 1 and Figure 2 shows schematically the idea behind the co-centric. Figure 1 shows a plate and a typical cross section. The area centre of the cross sections is geometrically on top of the plate centre line. This is the way we carried out the modelling.

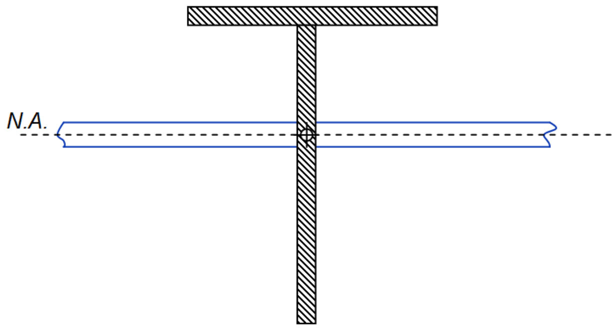


Figure 1 - Plate and beam as disconnected members. Plate and stiffener model example.

What is represented on Figure 1 is just the geometric modelling of the stiffened plate. In order to represent a structure as shown in Figure 2 is necessary to update the properties of the beam by being positioned at the top the plate in order to address the correct finite element model.

Thus, we keep the beams cross sectional area but an update computation of the inertia must be considered in order to account with the increment of the inertia given by the attached plate. The length of the attached plate should be introduced by the user (*s* parameter on Figure 2).

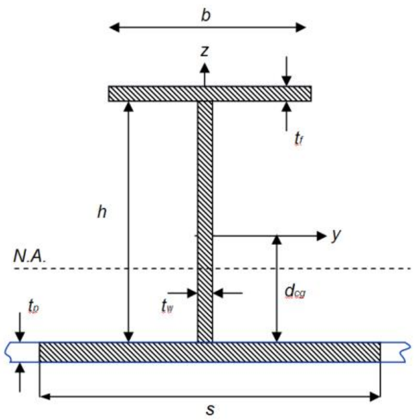
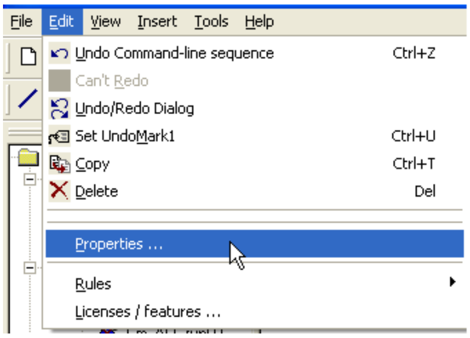
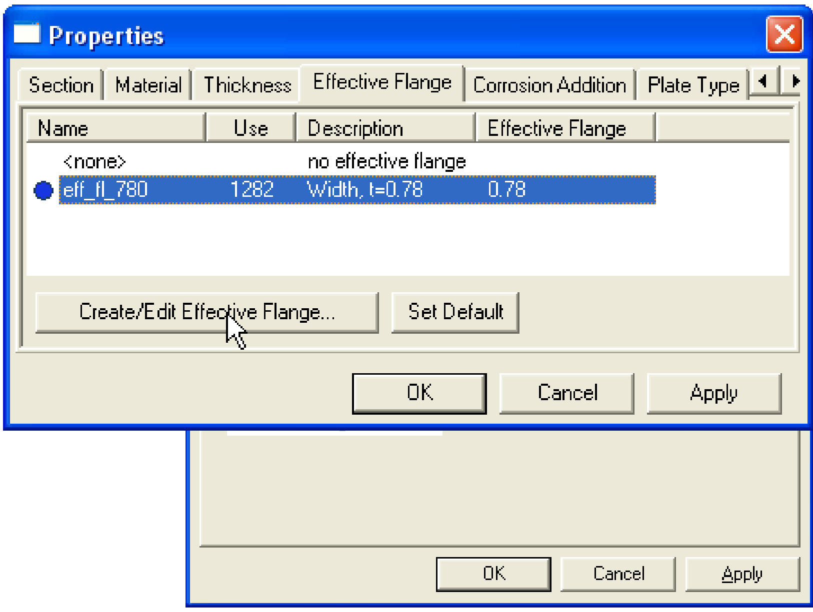


Figure 2 - Overall view of the beam properties. Plate and beam connected.

Corrosion addition is deducted from the stiffener profile and the effective flange prior to calculating the equivalent general section.

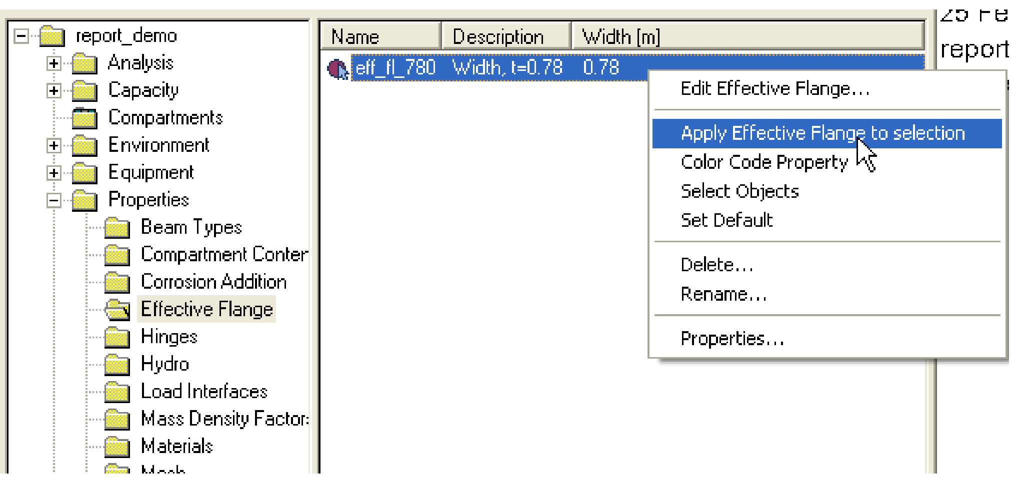
6.3.2 Using co-sentric stiffeners in Genie, Effective Flange





You can access the "Create/Edit Effective Flange" dialog by clicking *Edit|Properties* and select "Effective Flange" in the "Properties" dialog.

To apply the effective flange to beams in your model, you select the effective flange of your choice and click **RMB.**

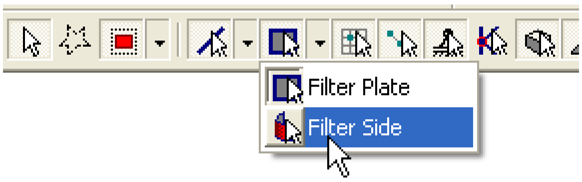


In the menu you choose *Apply Effective Flange to selection* to apply the chosen effective flange to your current selection or *Set Default* to use the selected effective flange on all new beams.

6.4 Some useful hints

6.4.1 Colour coding corrosion addition on plates

To be able to colour code or label the corrosion addition for plates you need to be able to select the sides of a plate. Select “Filter Side” in the top toolbar to colour code corrosion addition on plates.



6.4.2 Splitting up your model in smaller parts, limitation

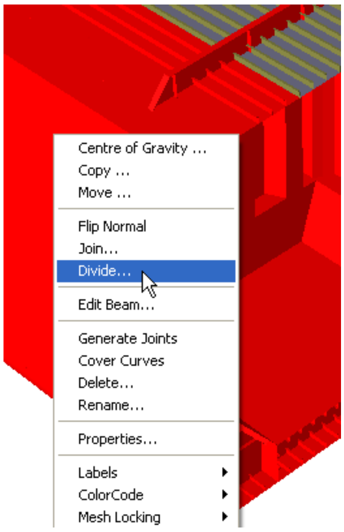
Sometimes you want to split up your model into smaller parts to in order to assign properties for smaller entities, or to create capacity models for a smaller part of the model-

To divide your model into smaller parts, select the plates or beams that you want to split click **RMB** and select *Divide*.

If you are working on a large cargohold model, you should not try to split the entire model in one operation. Such a big operation could actually make GeniE crash.

If you want to split up a large model you should do so in several smaller operations and save your work between each operation.

Further, it is better to do this operation before creating the compartment manager. Note that data exhange with Nauticus Hull must be performed once more after this kind of modifications is performed on the model.

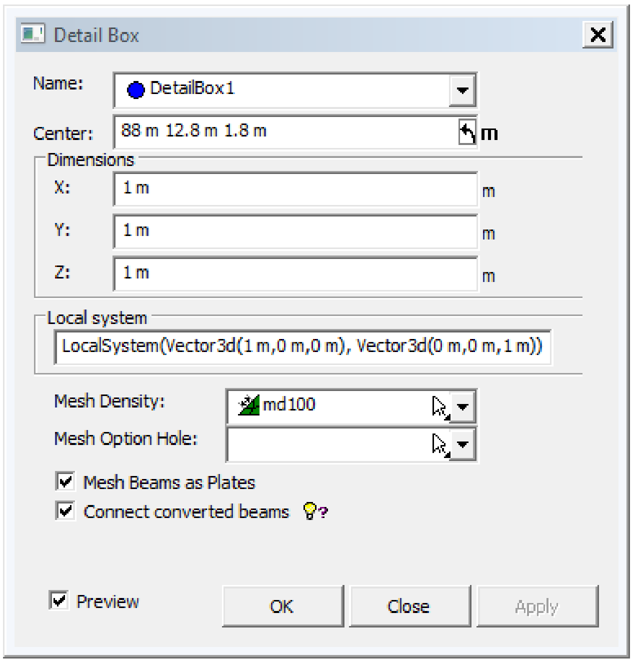


7 FATIGUE DETAIL EXPORT

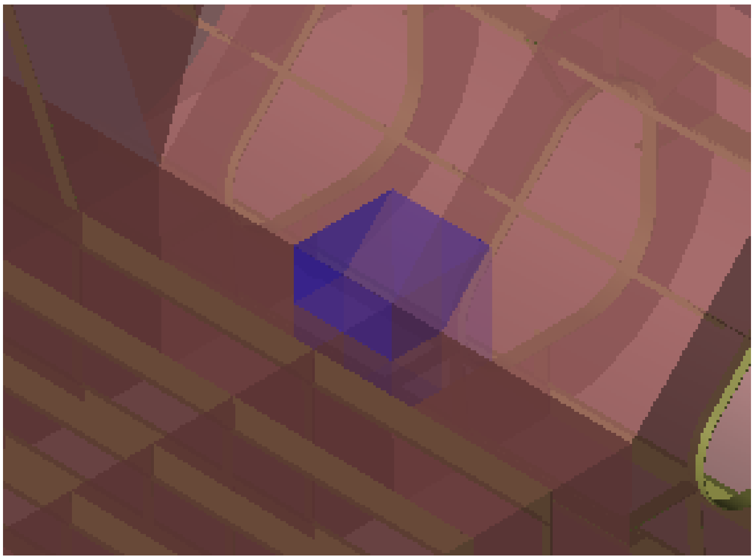
From a fine mesh analysis, relevant data for a fatigue check according to CSR BC&OT or DNV GL rules may be exported from GeniE.

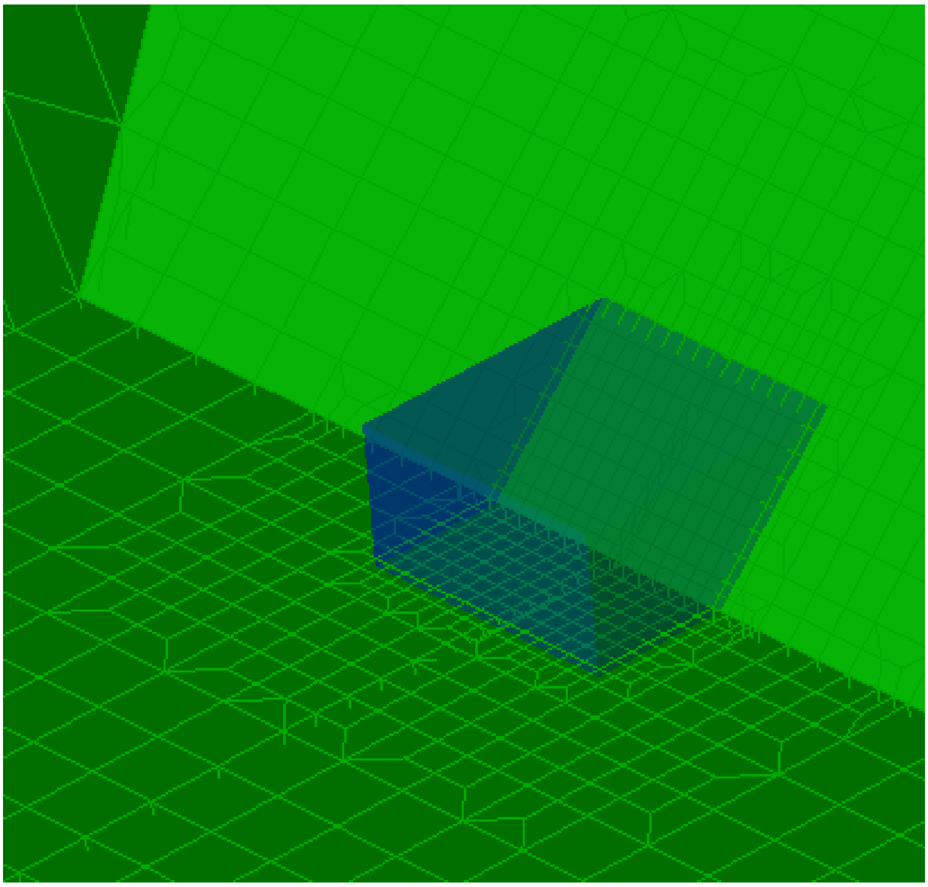
Execution of the Fatigue tool is not invoked from GeniE, but from the NH FEA template.

Use a Detail Box concept to identify the region of interest.



Defining a detail box

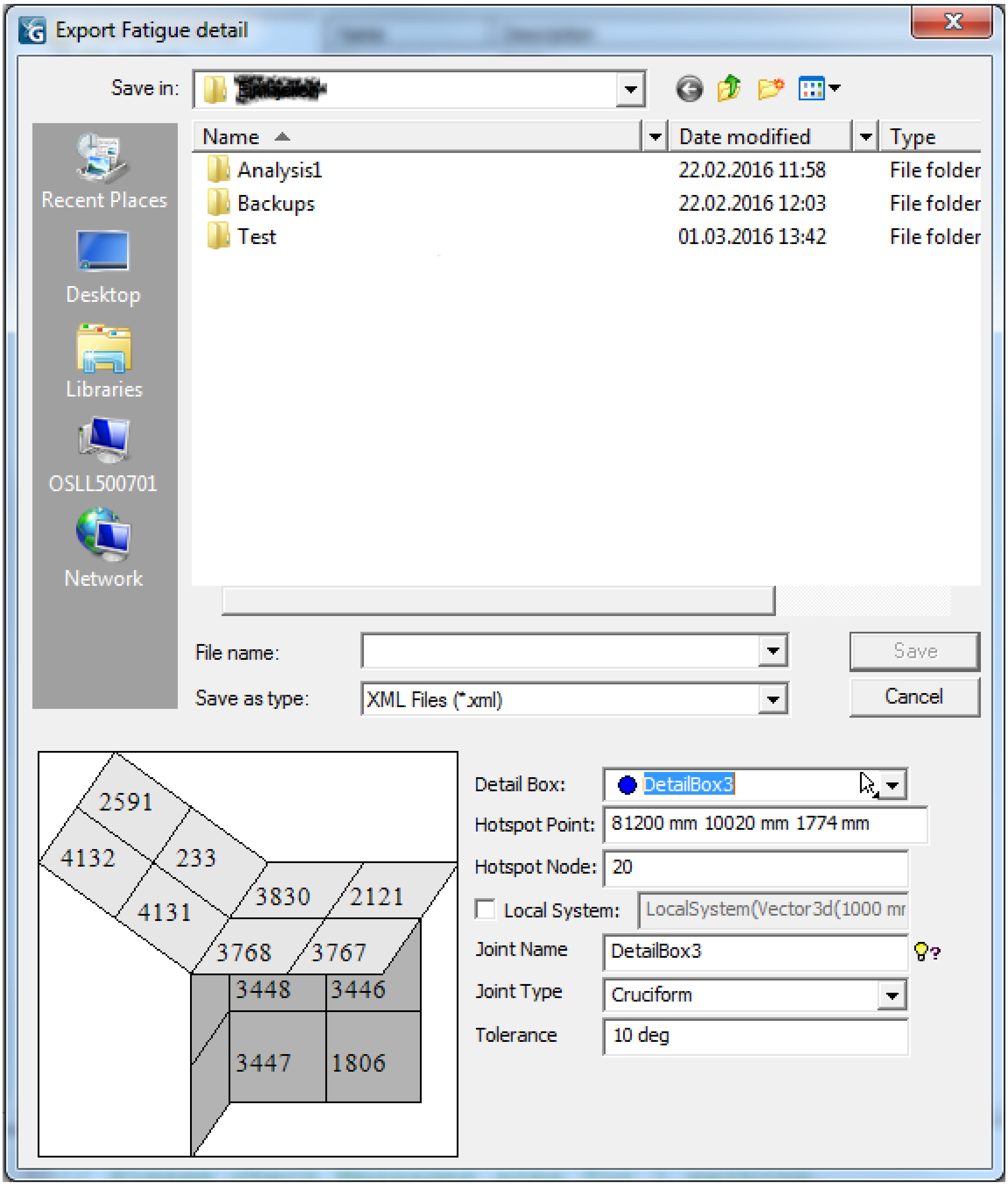




Fine mesh around detail box.

Export Fatigue Detail is available from the RMB menu when selecting a Detail Box

.



The purpose of the export fatigue detail is to export details from Genie to be analyzed in Nauticus Hull.

Detail Box: We define the hotspot you want to check by a Detail Box. You can override this by specifying any node in the Hotspot Node edit box.

Hotspot Point: We get this from the center of the Detail Box, but the user is free to give any point in the model. The program will use this point to extract a node to be used as Hotspot Node.

Hotspot Node: This is the node we will use as a basis for detecting fatigue detail models. You can change this as you want.

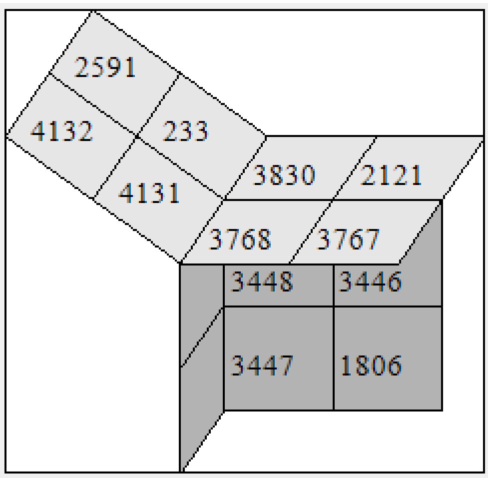
Local System: The program will try to autodetect the local system for the Joint Type you have selected, but you can manually override this if you do not agree with what the program selects.

Joint Name: This is the name that will be given to this fatigue detail when exported to XML.

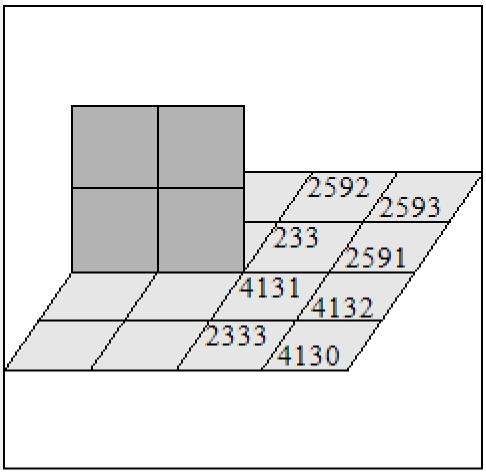
Joint Type: We currently support three joint types:

- Lower Hopper Knuckle (as seen on the picture 1 below)
- Cruciform (generalized HopperKnuckle with multiple webs)
- Type a (as seen on the picture 2 below)

Tolerance: Tolerance used during detection of the fatigue detail, typically in checking for parallel or orthogonal lines.



(1) Lower Hopper Knuckle



(2) Type a

8 APPENDIX 1 – COPYRIGHT NOTICE HDF5

GeniE uses the file format HDF5 to store info about the panels and results when using PULS.

8.1 Copyright Notice and License Terms for HDF5 (Hierarchical Data Format 5) Software Library and Utilities