SESAM TUTORIAL

GeniE
Member Code Checking

Valid from program version 8.2

Sesam Tutorial

GeniE – Member Code Checking

Date: June 2021

Valid from GeniE version 8.2

Prepared by: Digital Solutions at DNV

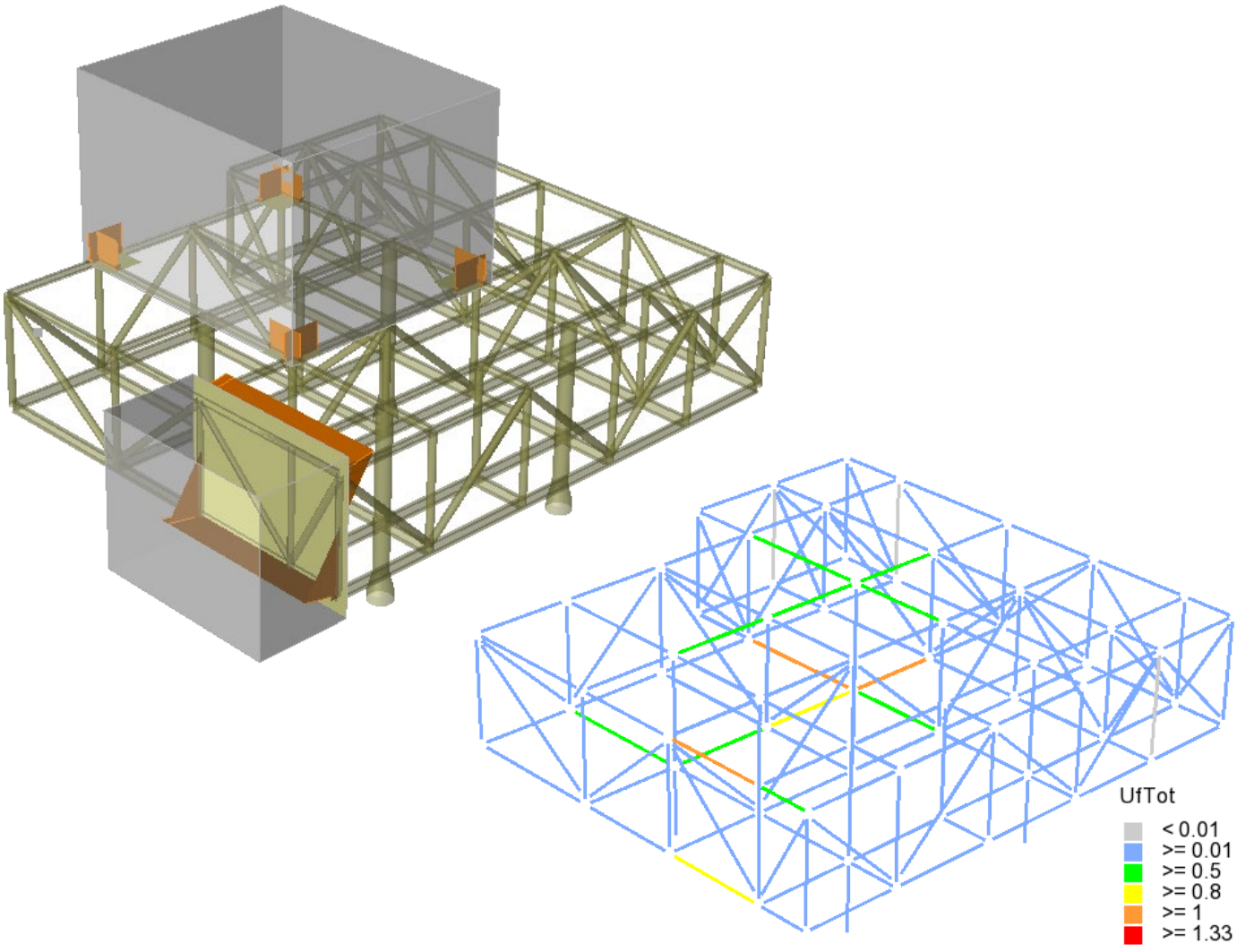
E-mail support: software.support@dnv.com

E-mail sales: digital@dnv.com

TABLE OF CONTENTS

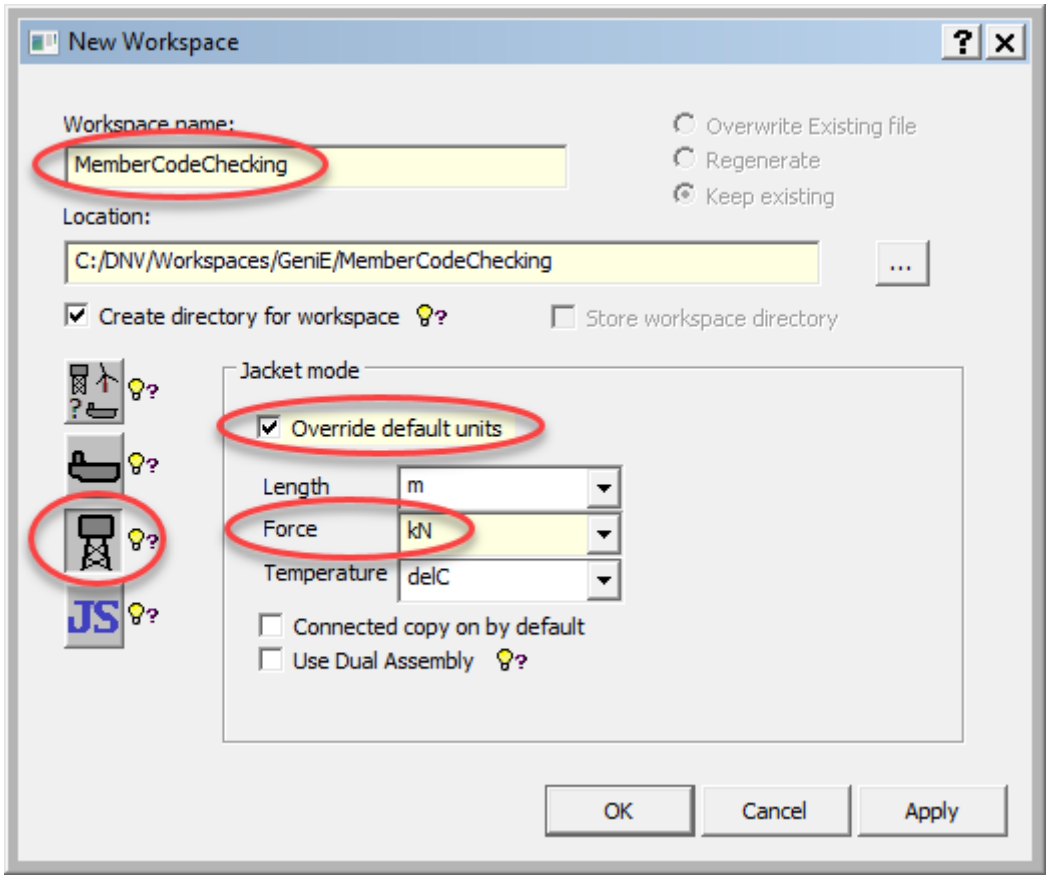
1 INTRODUCTION

- In this tutorial member code checking according to API WSD is performed for the topside model created in tutorial ‘B5 GeniE Topside’. Said tutorial includes first a pure beam modelling of the topside and secondly a detailed plate/shell modelling of a selected joint. The present tutorial is based on the pure beam model.
- This tutorial contains the following steps:
  - Open a new workspace and import the model of tutorial ‘B5 GeniE Topside’.
  - Create a *capacity manager*, fill this with *capacity members* and add a *run*.
  - Generate *code check loads* and *execute code checking*.
  - Redesign failing members.
  - Rerun static analysis and code check after redesigning failing members.
  - Report code check results.
- The model with equipments and code checking results are displayed below.



2 OPEN NEW WORKSPACE AND IMPORT MODEL

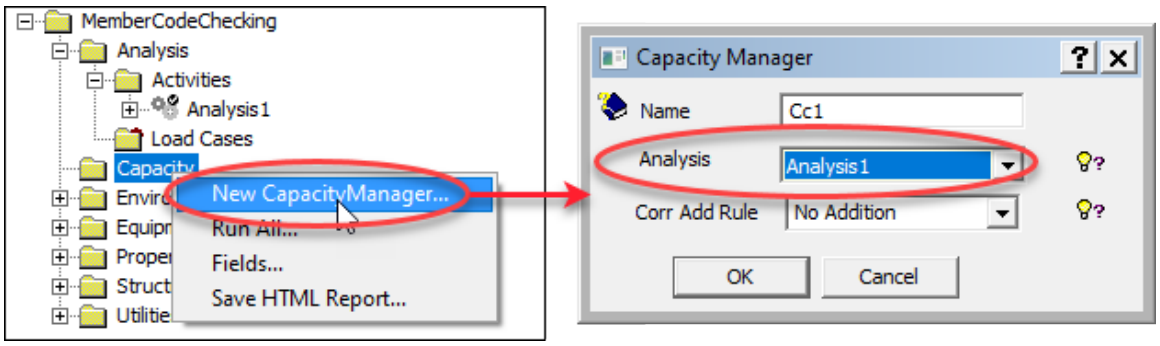
- Start GeniE and open a new workspace.
  - Give a *Workspace name*.
  - Click the *Jacket mode* button to customise for jacket (frame) modelling, i.e. limit menus and buttons to those relevant for frame modelling.
  - Set *Force* unit to kN and click *OK*.



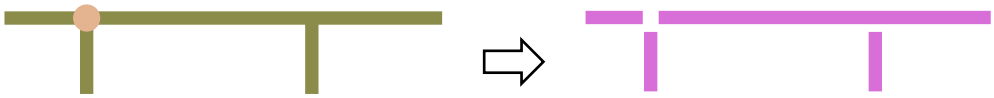
  - If the workspace name exists select *Overwrite Existing file* or give another name.
- Use *File* | *Read Command File* to read the js file of tutorial 'B5 GeniE Topside' named Topside_input.js. Tutorials are found in the installation folder typically named <path>\GeniE VX.Y-ZZ\Help\Tutorials\.
  - See that a static analysis is run as the last step of this js file.
  - Switch to 'Modelling - Transparent' display configuration.
  - The imported model contains three load cases: gravity, helideck equipment loads, flare tower equipment loads and a combination of these three. The combination named LC_total has been assigned operating design condition. This can be seen by right-clicking the combination found in folder *Analysis* | *Load Cases*, selecting *Properties* and going to the *Design Condition* tab.

3 CREATE CAPACITY MANAGER AND FILL WITH CAPACITY MEMBERS

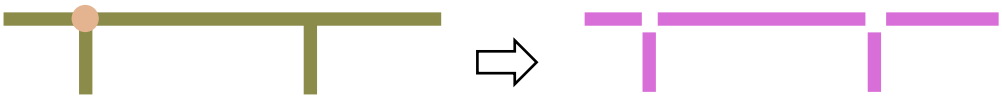
- Right-click the *Capacity* folder in the browser to create a capacity manager named e.g. Cc1. Select the recently run static analysis named Analysis1 and click *OK*.



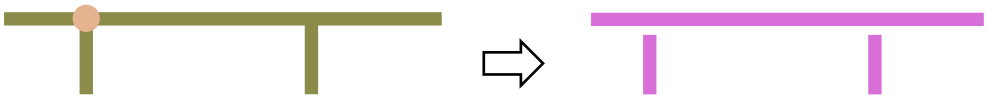
- Right-click the capacity manager to fill it with capacity members. In the *Create (Update) Members* dialog select the option *Split at incoming beam*. A few comments regarding the other options in the dialog:
  - *Subset* allows a set rather than the complete model to be code checked. This could be checked to select for example the set Main_deck. In this tutorial, code check the complete model.
  - *Split at joint* involves splitting at joints when creating members from beams.

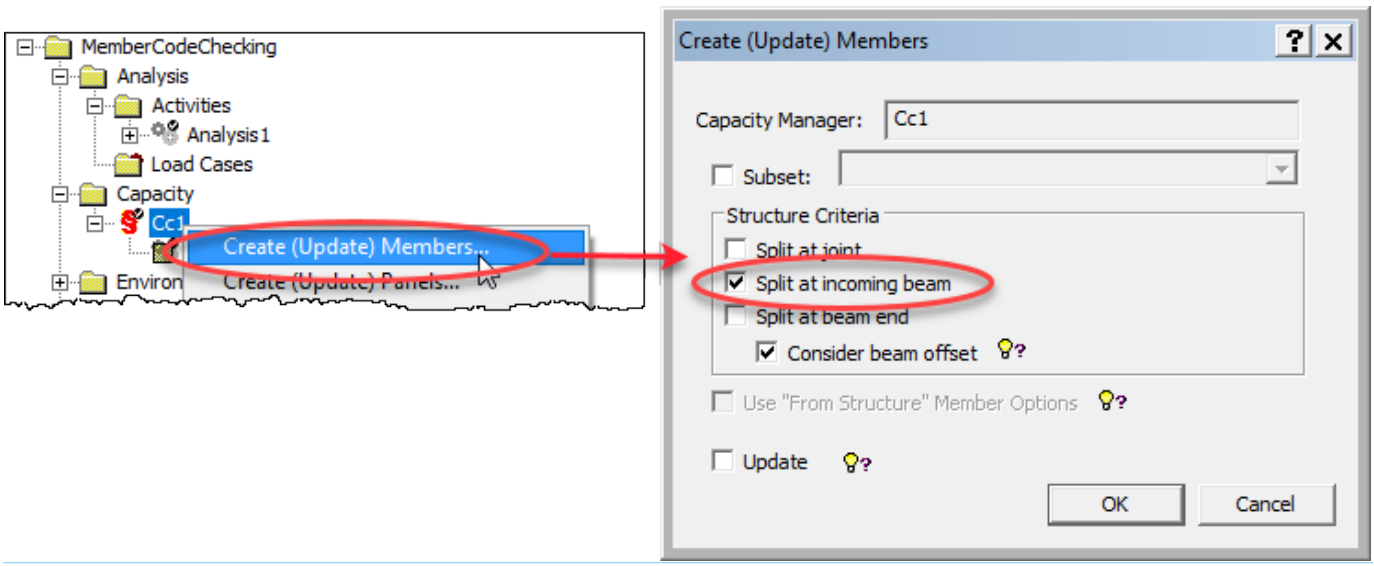


  - *Split at incoming beam* involves splitting at all beam intersections.



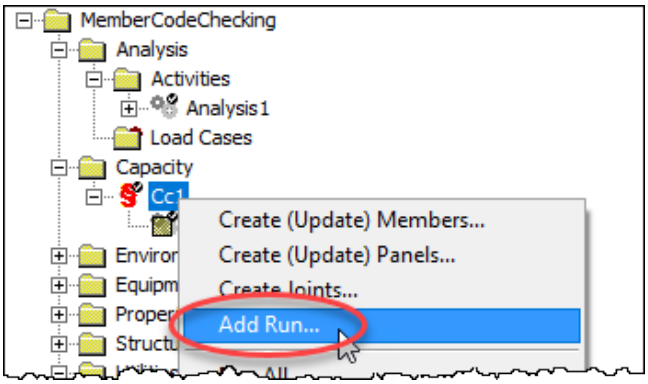
  - *Split at beam end* involves no splitting of beams when creating members.



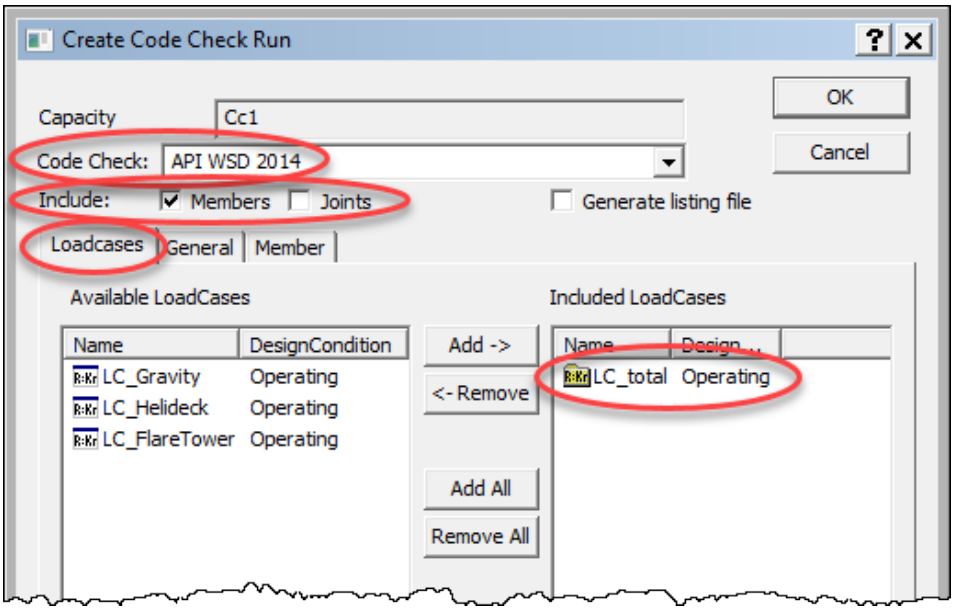


4 ADD RUN

➢ Right-click the capacity manager and select *Add Run* to open the *Create Code Check Run* dialog.

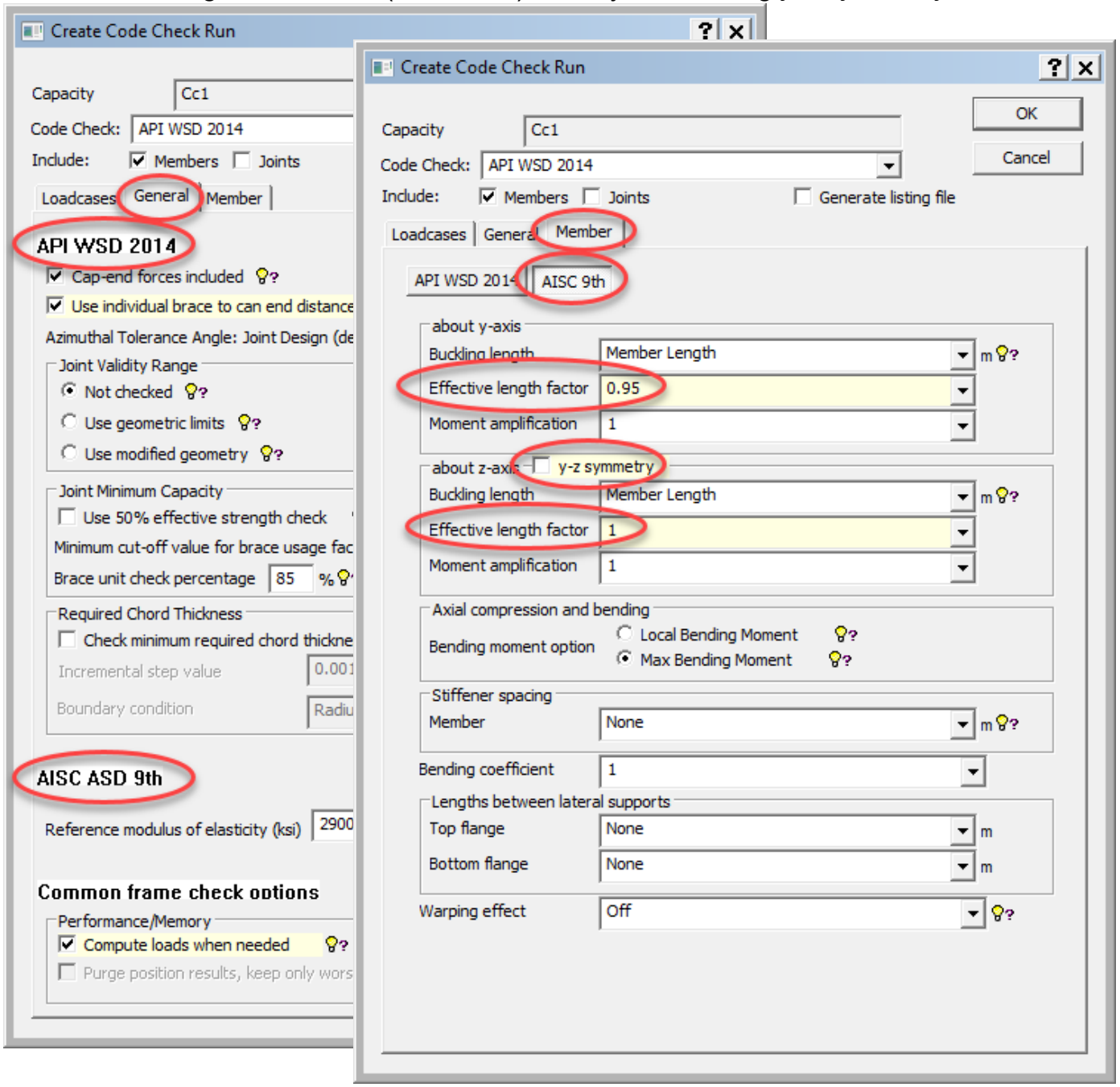


- In the dialog select *Code Check API WSD 2014*.
- Leave *Members* checked but uncheck *Joints* as tubular joints will not be code checked in this tutorial.
- In the *Loadcases* tab, select the load combination LC_total listed in the *Available LoadCases* field and click *Add* to move it to the *Included LoadCases* field. I.e. code check only the combination.

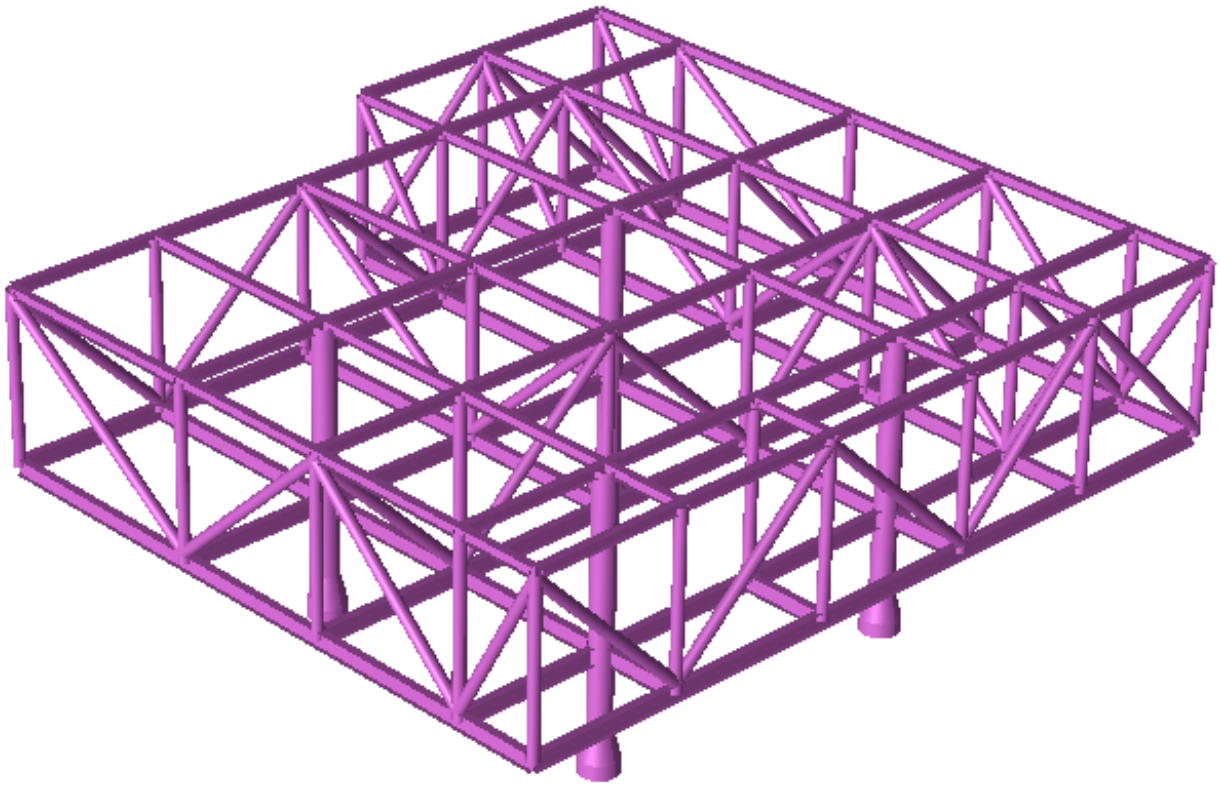


- Don't *OK* the dialog yet as the next page deals with the *General* and *Member* tabs of the *Create Code Check Run* dialog.

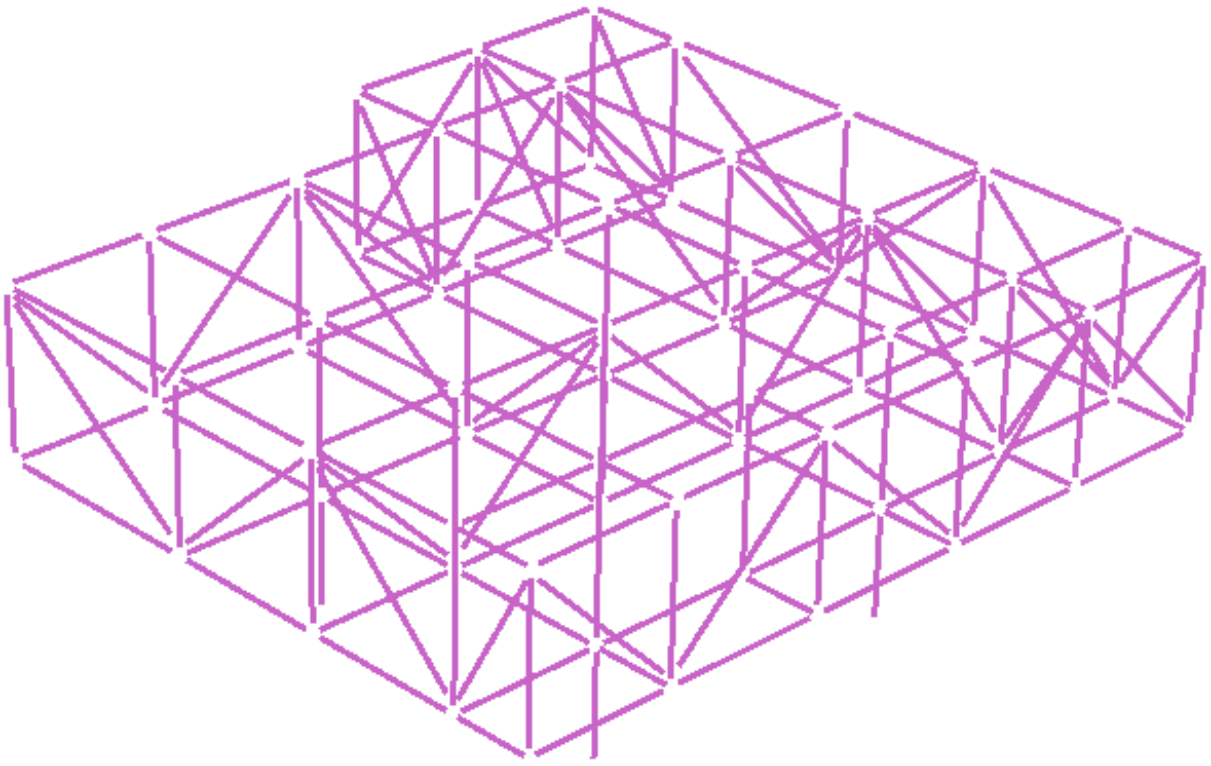
- In the *General* tab some general options may be modified. In this tutorial accept the default settings. Notice that the dialog informs that while tubular members are checked according to *API WSD 2014*, non-tubulars are checked according to *AISC ASD 9th*.
- In the *Member* tab *Buckling length*, *Effective length factor*, *Moment amplification* and other data may be given. Note that these data are valid for *all* members. These data are modified for selected members as explained later.
  - Click the *AISC 9th* tab (non-tubulars) and give *Effective length factor* 0.95 for buckling *about y-axis* (strong axis) for all members. Leave the factor for buckling *about z-axis* (weak axis) as 1 by unchecking *y-z-symmetry*.



➢ Having clicked *OK* in the *Create Code Check Run* dialog, switch to *Capacity Models* to display the capacity members.

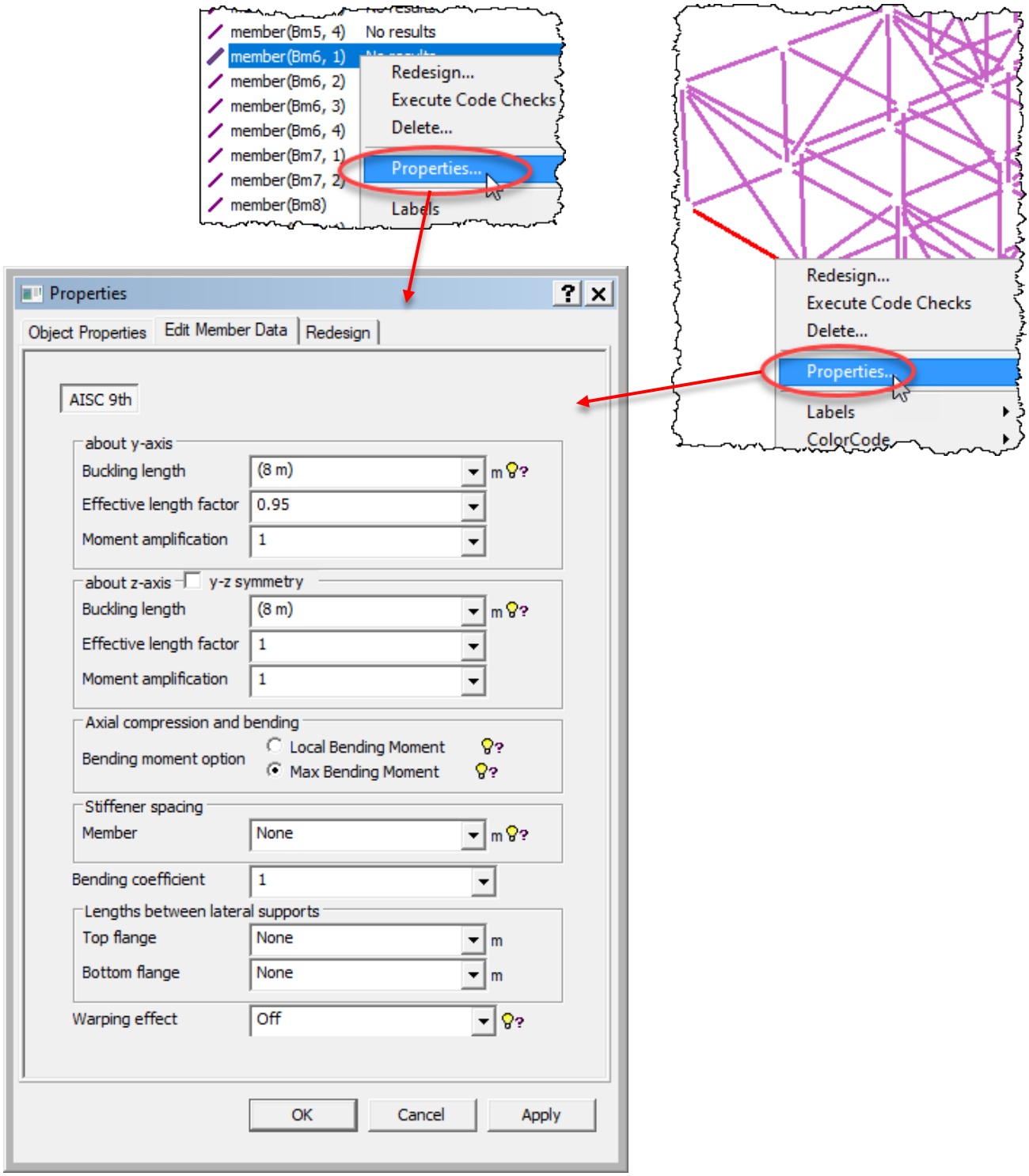


- Switch to *Wireframe* view ( ) to see the individual capacity members.



5 CHANGE BUCKLING DATA FOR SELECTED MEMBER

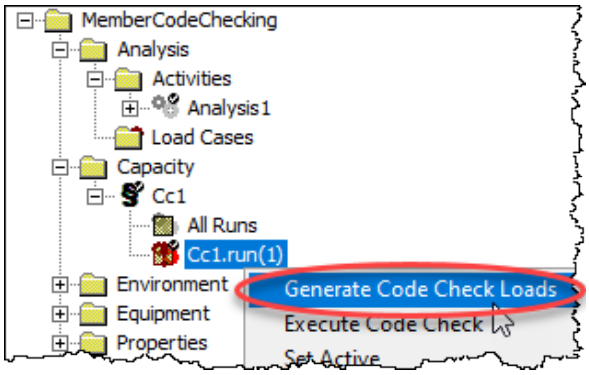
➢ The buckling data may be modified for individual capacity members by right-clicking them in the browser or in the display.

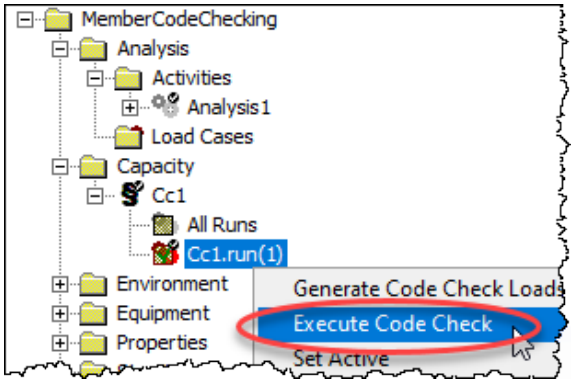


- However, in this tutorial do not modify buckling data for individual members.

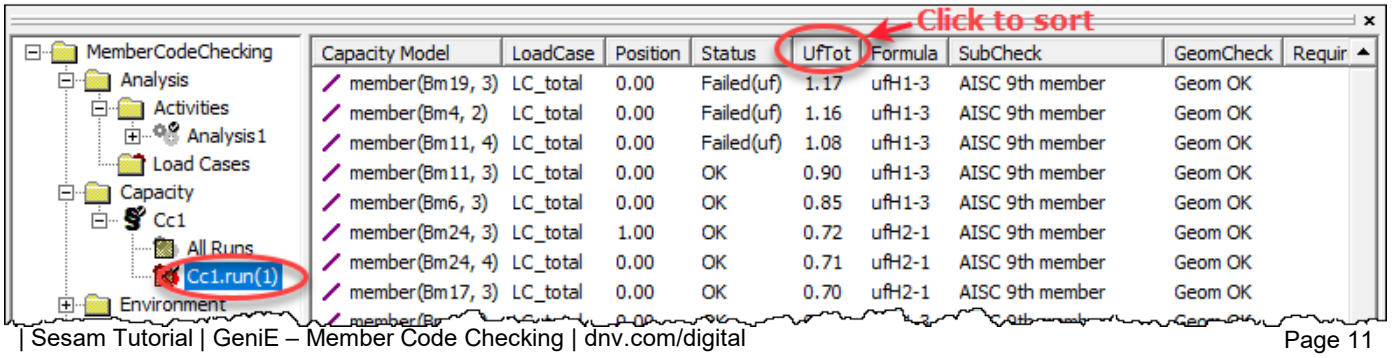
6 GENERATE CODE CHECK LOADS AND EXECUTE CODE CHECKS

- Right-click the code check run and click *Generate Code Check Loads*. For a big model this may require a few seconds.
  - The FE analysis in Sestra computes forces and moments at the ends and midpoints of beam elements. However, code checking must also be done at intermediate position where the forces and/or moments may be high, e.g. due to local beam loads. The *Generate Code Check Loads* process, therefore, loops over all load cases/combinations selected for code checking, determines positions for code checking and computes the forces and moments at these positions. Typically, these positions are:
    - At beam ends and quarter positions (0, 0.25, 0.5, 0.75, 1)
    - Where the cross section and/or material properties change
    - At positions of maximum in-plane and out-of-plane moments
- Right-click the code check run and click *Execute Code Check* to do the actual code checking. Again, for a big model with many load cases/combinations selected for code checking this may require a few seconds.

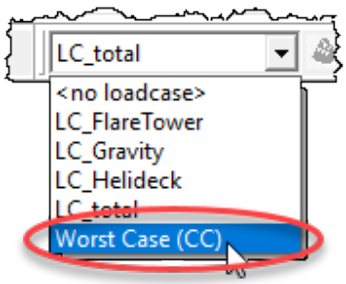




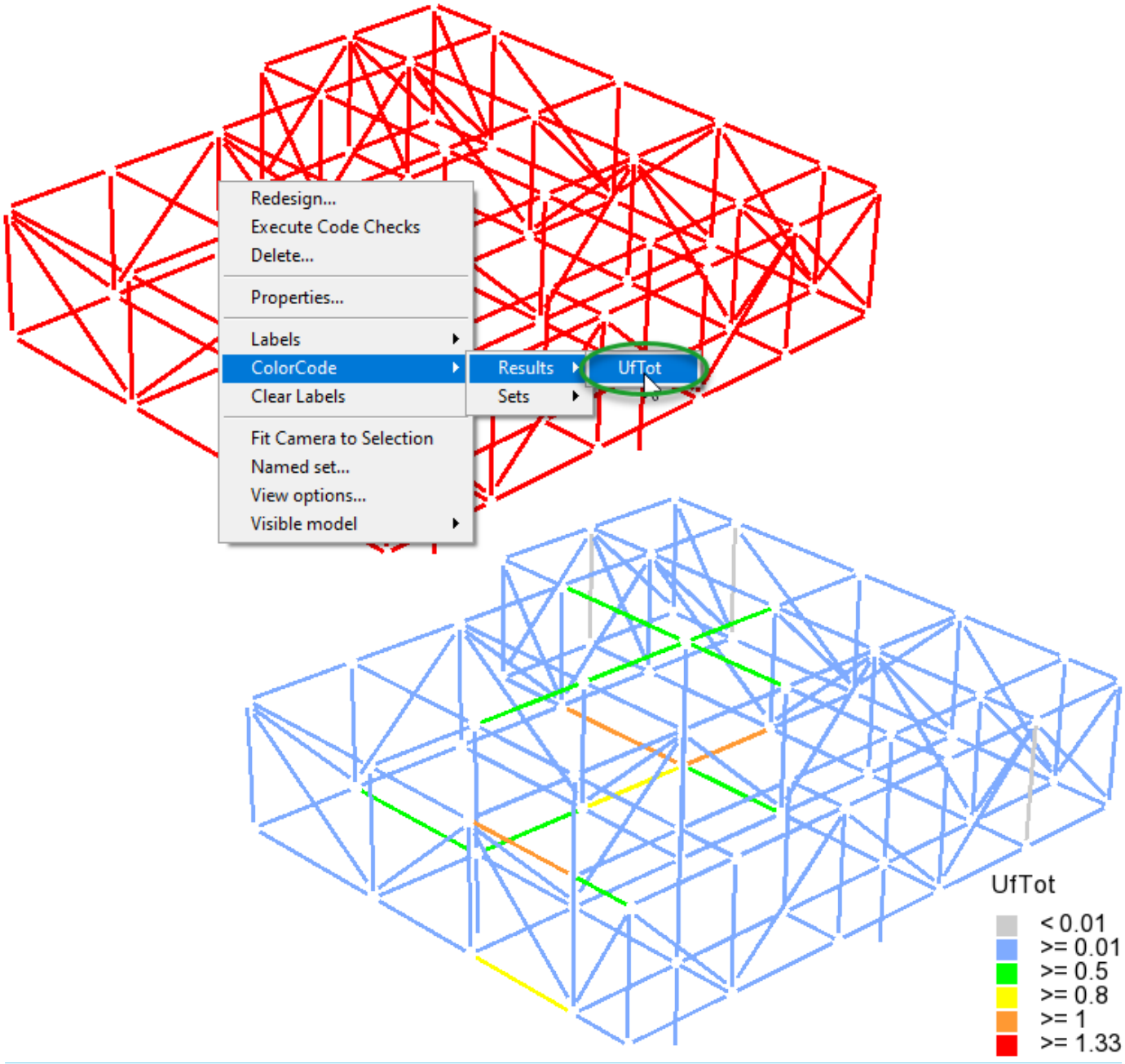
- The browser now tabulates the code check results. For each capacity member (the name refers to concept beam and capacity member number within it), the most severe load case/combination and position along it [0, 1] is listed. *Status Failed(uf)* means that the member fails, and the usage factor is given in the next column *UfTot*, the highest of several usage factors computed. The *Formula* column contains references to specific equations in the code (standard) as explained in 'Reference Documents' found on the GeniE help page (*Help* | *Help Topics* or F1). For example, *ufH1-3* means 'Usage factor according to section H1-3' in AISC 9th.



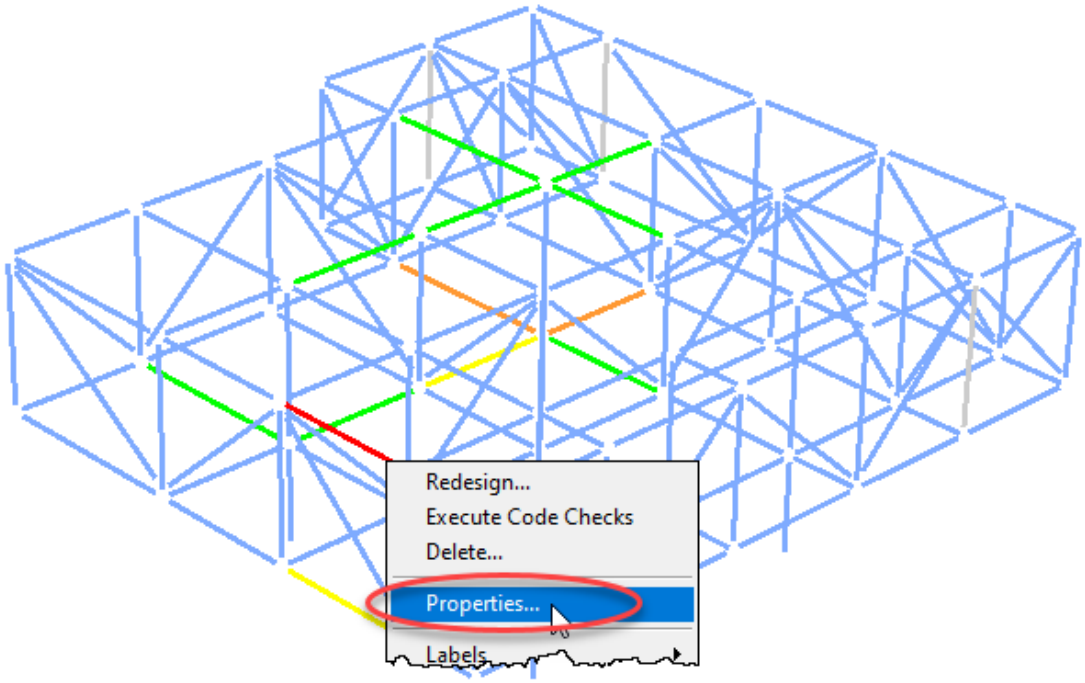
- The code check results tabulated and displayed (see below) are for the currently selected load case/combination. When several cases/combinations have been code checked use the load case selector to select *Worst Case (CC)*.



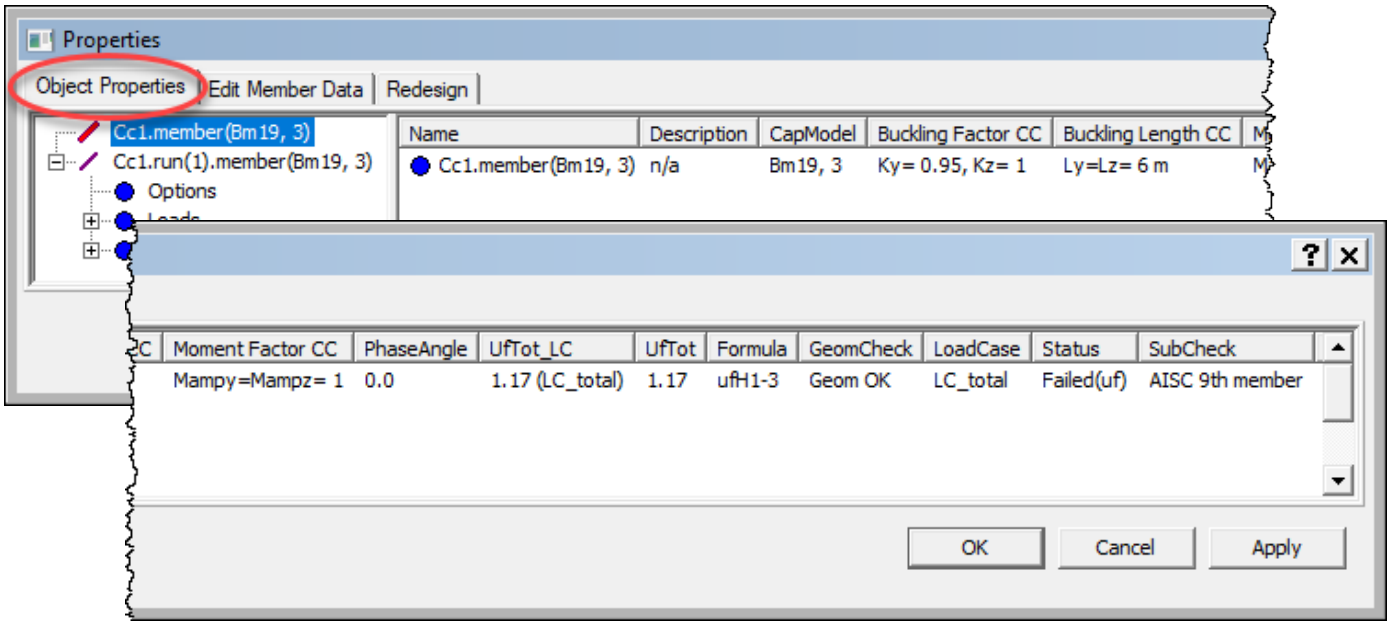
- Display the code check results by selecting the whole capacity model, right-clicking and selecting *ColorCode* | *Results* | *UfTot*. Then click outside the model to deselect it so as to see the colour coding.
  - Note that you may also right-click and select *Labels* | *Results* to label selected members with data like buckling length and factors, decisive code check equation, usage factor and more.



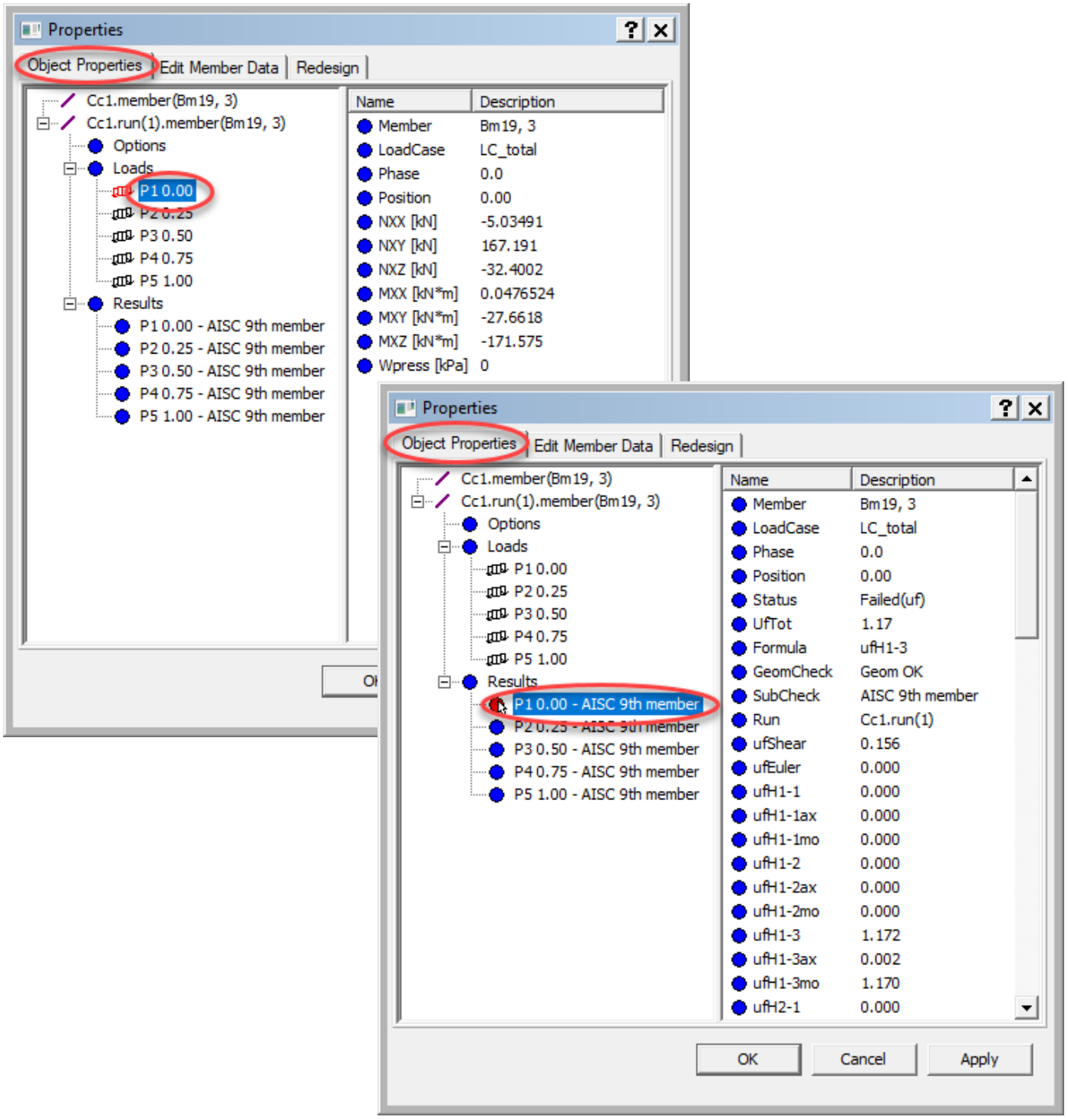
➢ See more details for a selected member by right-clicking and clicking *Properties*.



- In the *Object Properties* tab find buckling data and code check results for the member.



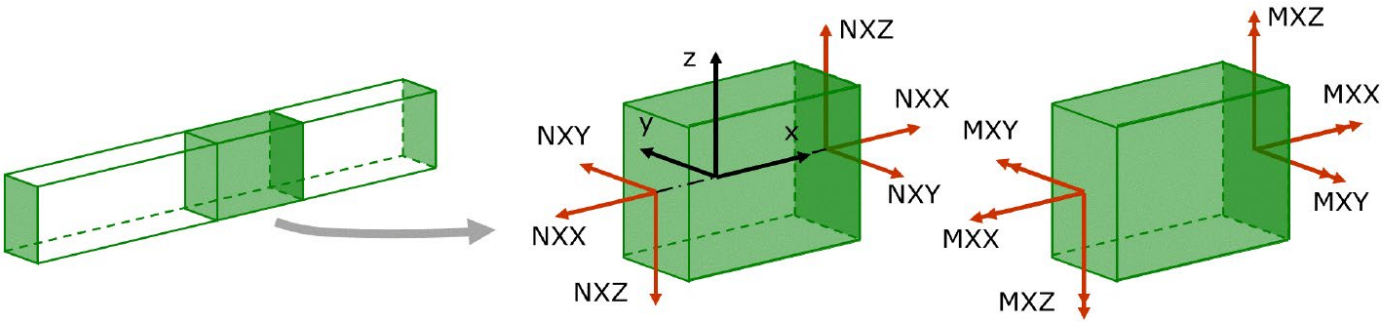
- Also in the *Object Properties* tab expand *Loads* and select a position to see the forces and moments at that position.
- Expand *Results* and select a position to see all code check results at that position. I.e. not only the highest usage factor *UfTot* but usage factors for any equation of relevance in the code.



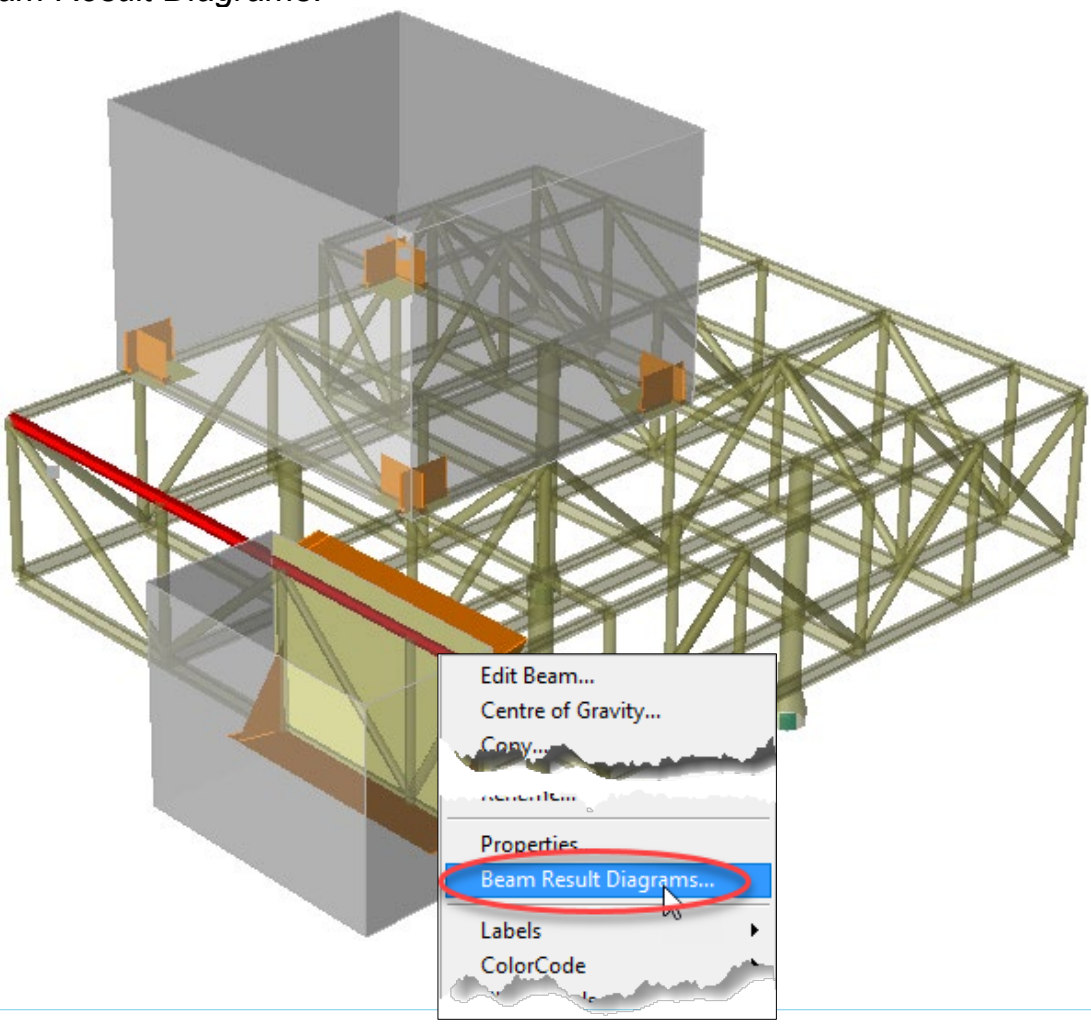
7 UNDERSTAND FORCES AND MOMENTS

➢ To increase your understanding of the reason for the failure of the member, return to *Modelling - Transparent* and select the combination LC_total.

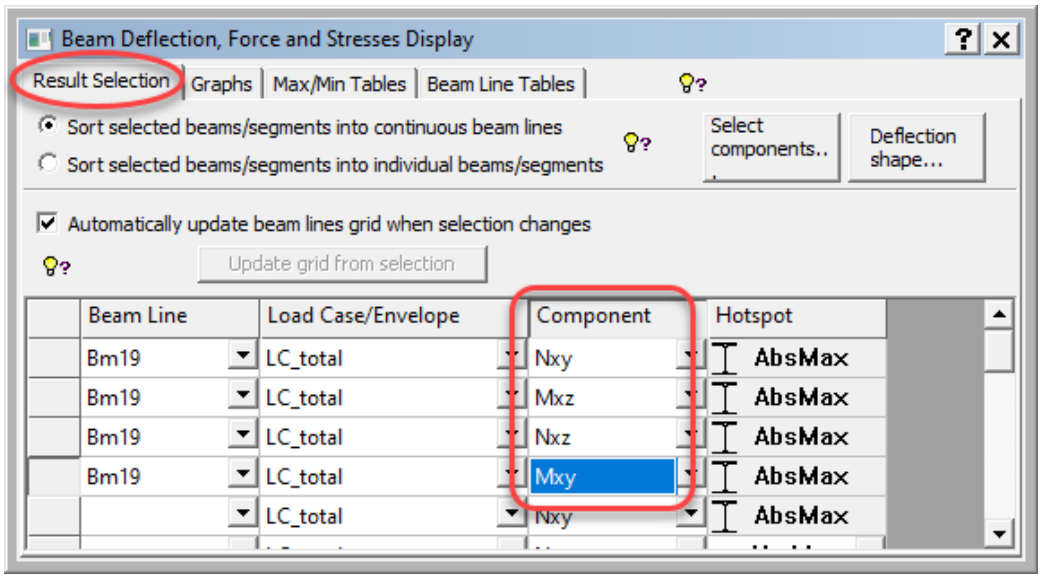
- The equipment hanging on the side of the deck is a flare tower. The line load shown in orange colour hit the concept beam highlighted in red.
- Since the horizontal component of the line load is larger than the vertical component the horizontal shear force N_{XY} will be larger than the vertical shear force N_{XZ}. And the moment about the vertical axis M_{XZ} will be larger than the moment about the horizontal axis M_{XY}.

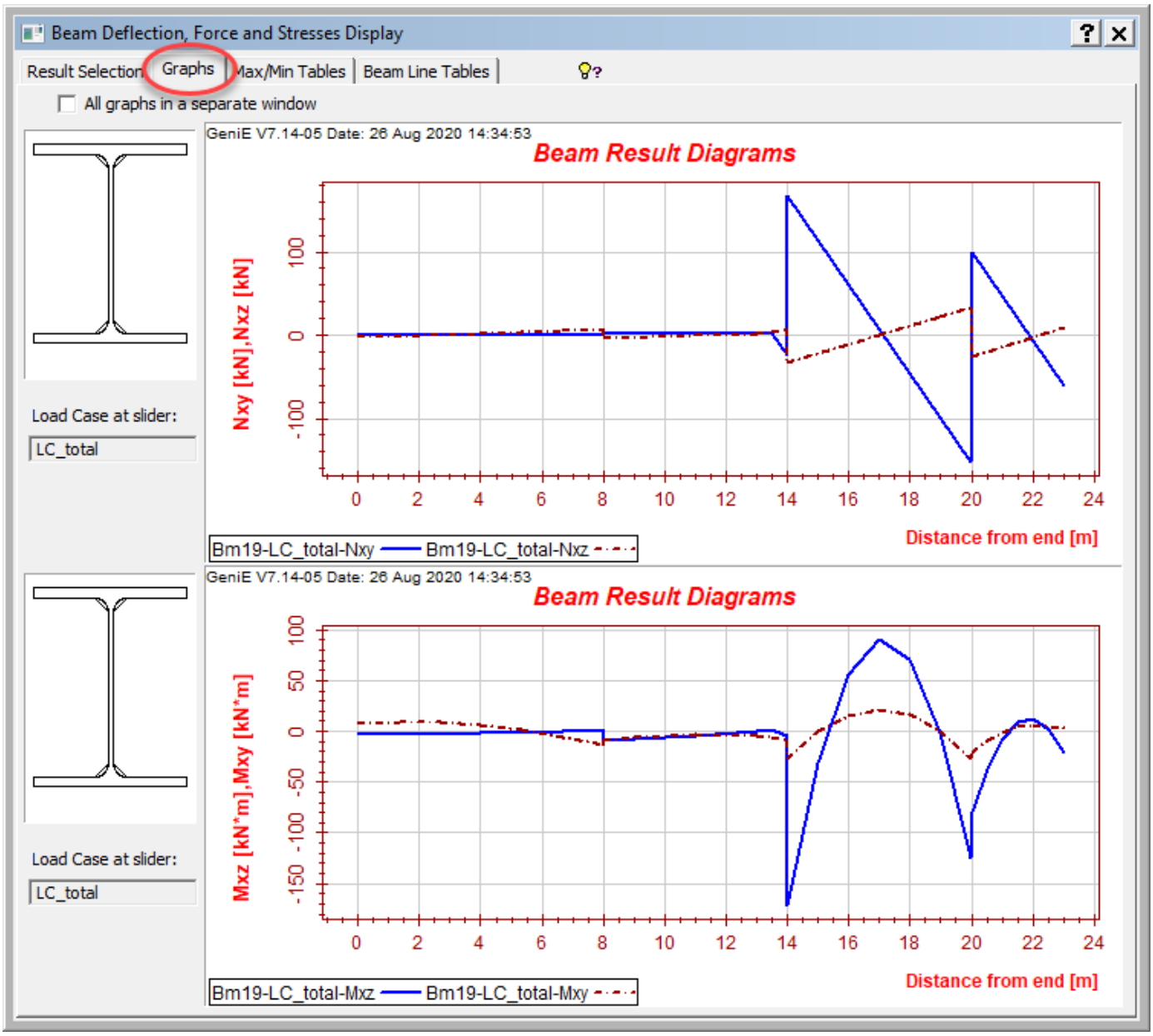


- Select the concept beam at the edge of the main deck, right-click and select *Beam Result Diagrams*.



- In the *Result Selection* tab, *Component* column, select the shear forces and moments, *Nxy*, *Mxz*, *Nxz* and *Mxy*.
- Then go to the *Graphs* tab to see the graphs.



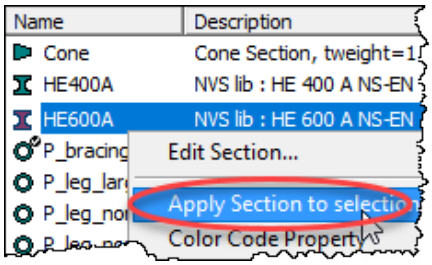


8 REDESIGN

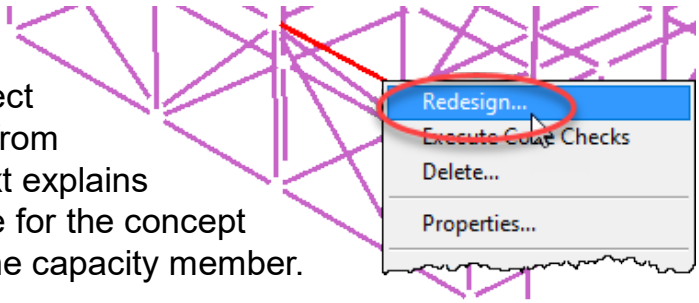
- The failing HE400A beam at the edge of the main deck must be redesigned, i.e. made stronger. There are alternative approaches for doing this redesign.

- **Modify Concept Beam Approach**

  - Select the *concept beam*, find the larger cross section HE600A in the browser and apply it to the beam, i.e. do not apply to the capacity member.

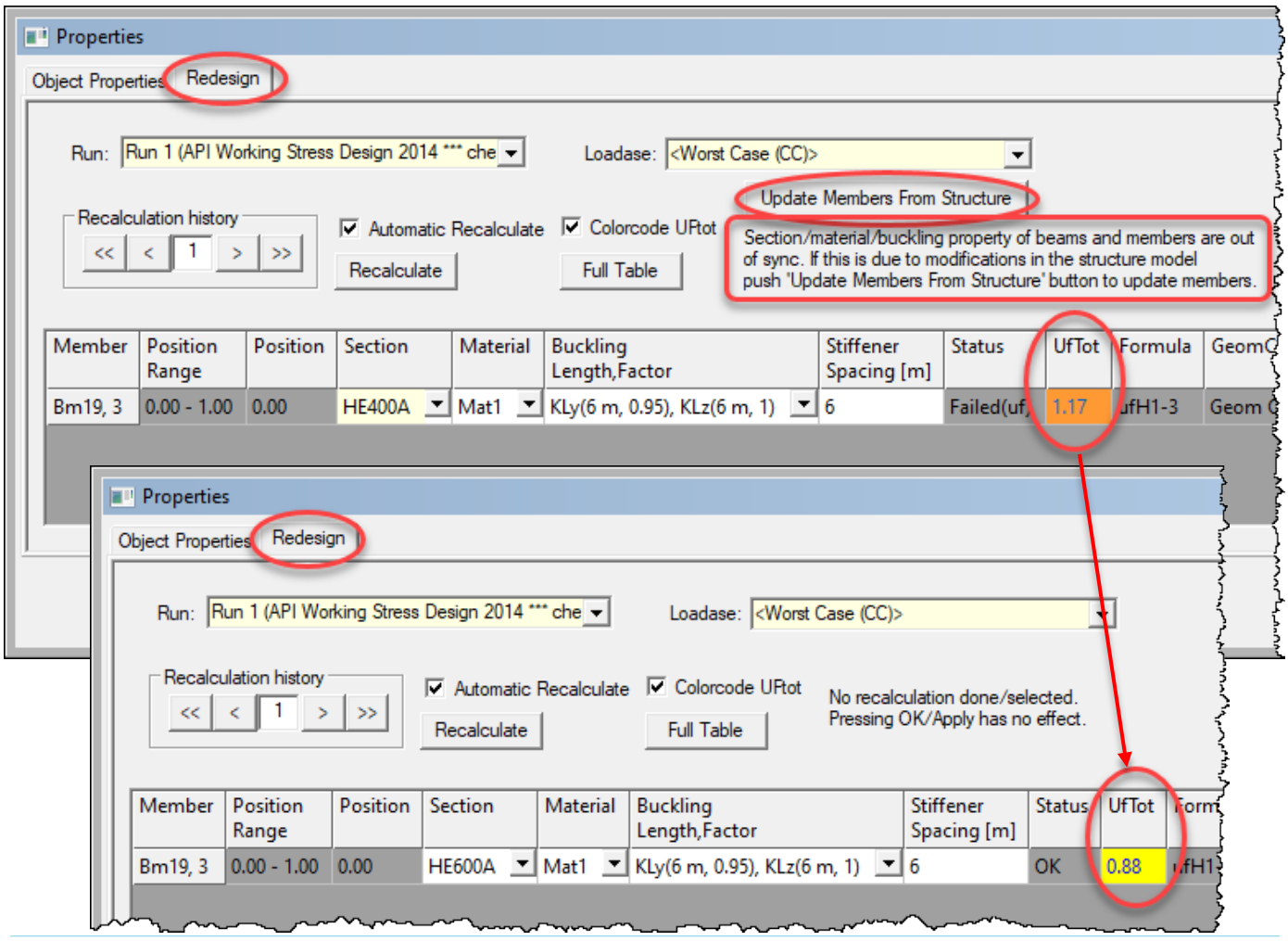


  - Switch to the *Capacity Model* display configuration, select the failing beam and right-click to select *Redesign*.

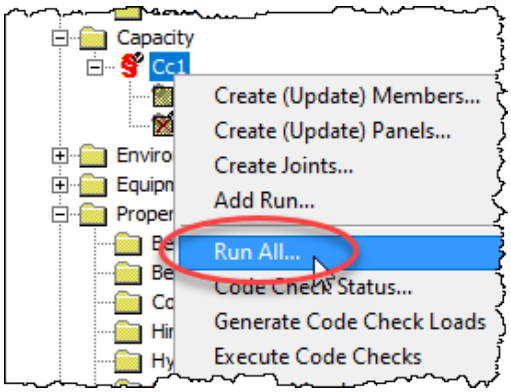


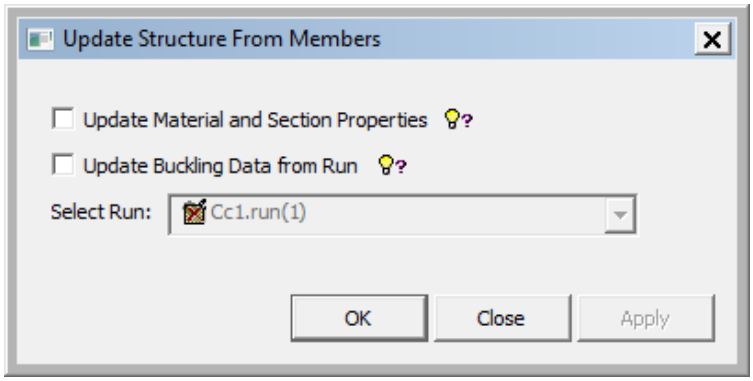
  - See that *UfTot* is as before, i.e. no effect yet from increasing the cross section from HE400A to HE600A. The encircled text explains why: the change of cross section done for the concept beam hasn't yet been transferred to the capacity member.

  - Click *Update Members from Structure* (update capacity members from concept beam) and see *UfTot* decreases.

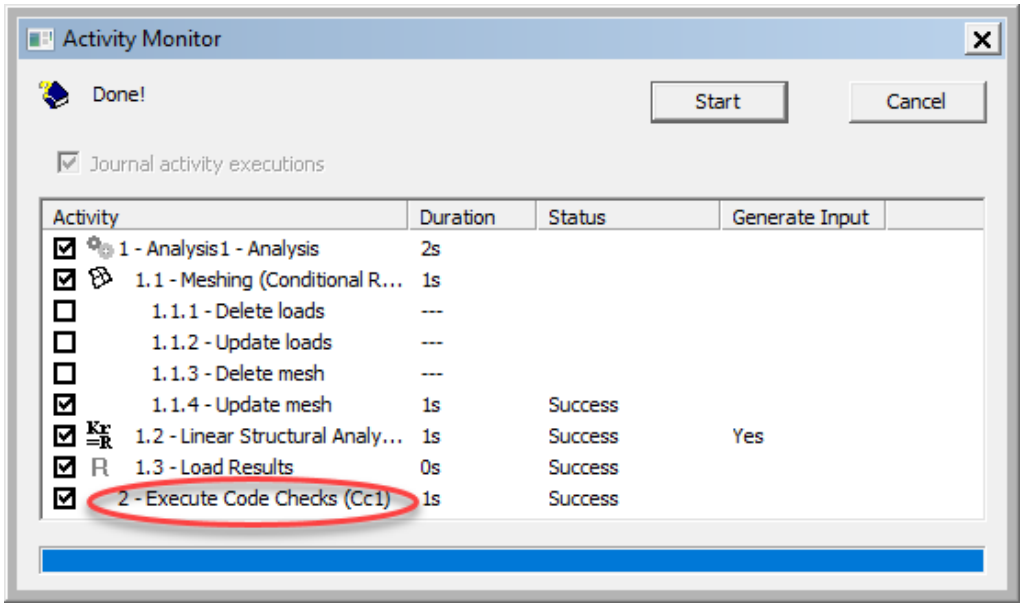


- However, this redesign was done without rerunning the static analysis. I.e. the redesign is based on the assumption that the increase of section from HE400A to HE600A will not change the forces and moments in the redesigned beam.
- This is of course not the case. The stronger cross section will involve a redistribution of loads in the deck structure and cause somewhat larger forces and moments in the redesigned beam.
- To remedy this, right-click the capacity manager in the browser, right-click and select *Run All*.
- The *Update Structure From Members* dialog appears. This is to transfer any modifications done to the capacity members back to the concept beams. No such modifications has been done at this point. Click *OK*.



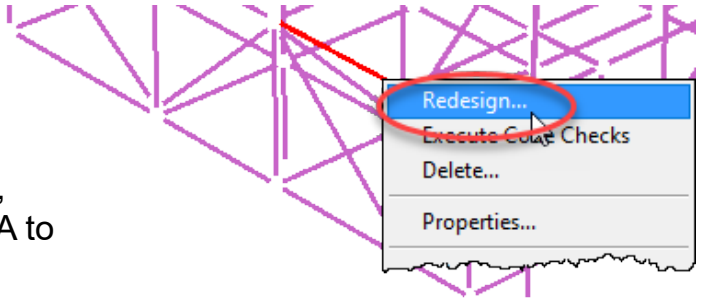


- The *Activity Monitor* appears. Click *Start* to run the static analysis anew plus executing the code check anew. The utilisation of the redesigned beam may increase somewhat after this step due to the redistribution of forces.

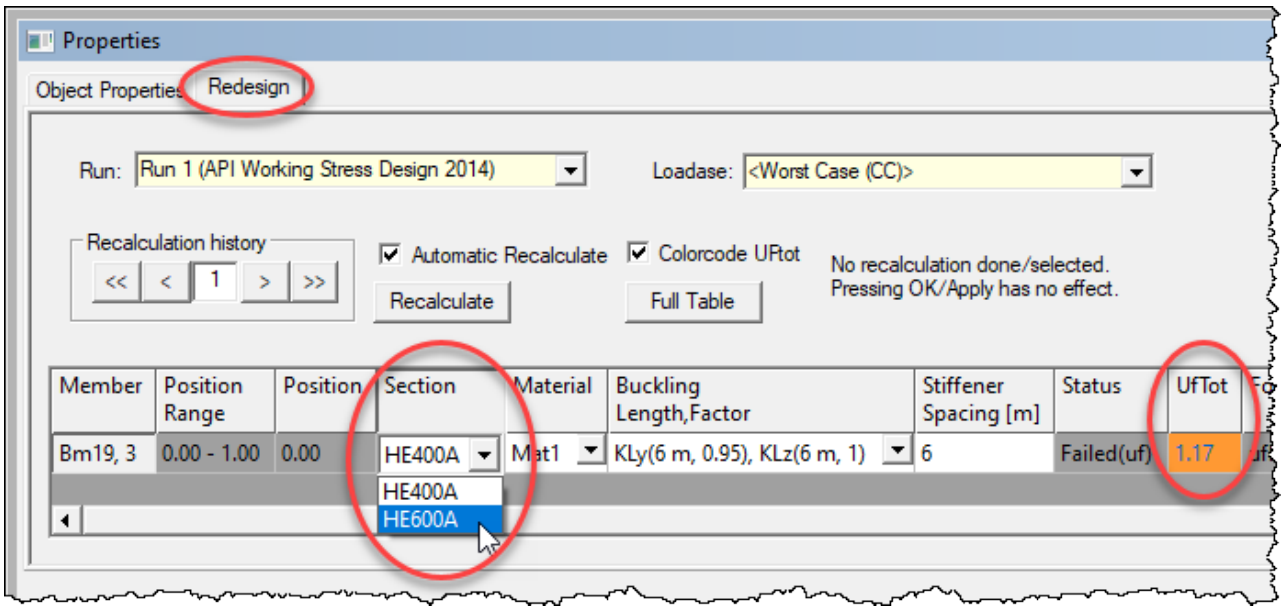


- Undo (Ctrl+Z) this change of cross section to try out another approach.

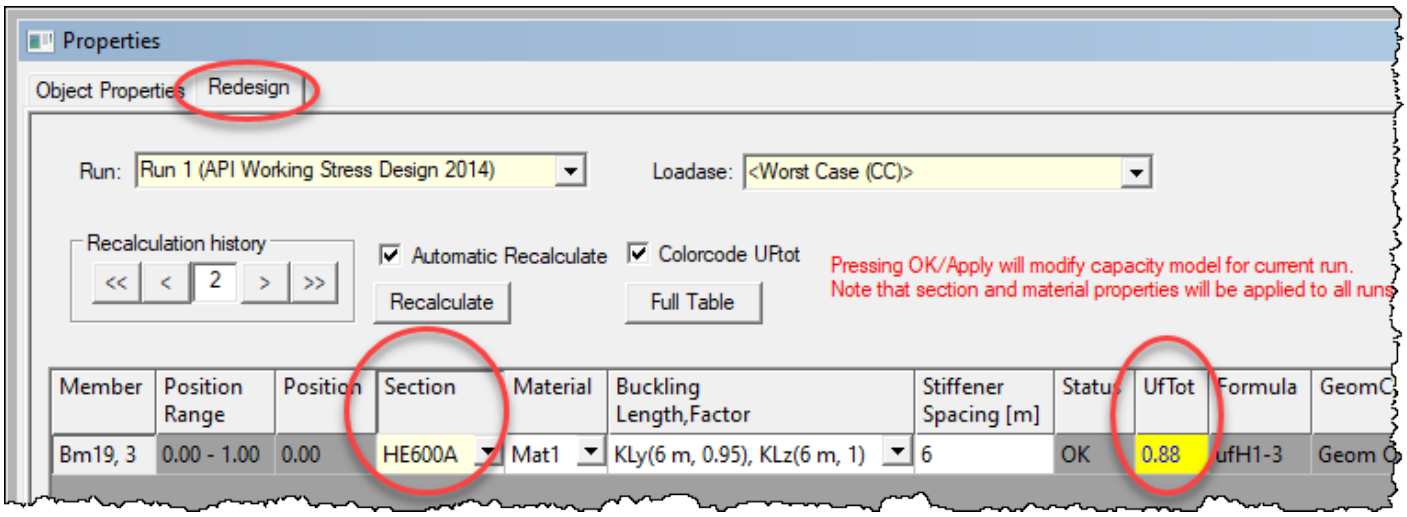
➢ Modify Capacity Member Approach



- Select the failing beam and right-click to select *Redesign*.
- In the *Properties* dialog, *Redesign* tab, change the cross section from HE400A to the larger cross section HE600A.
- As opposed to the modify concept beam approach, this change is done to the *capacity member*.

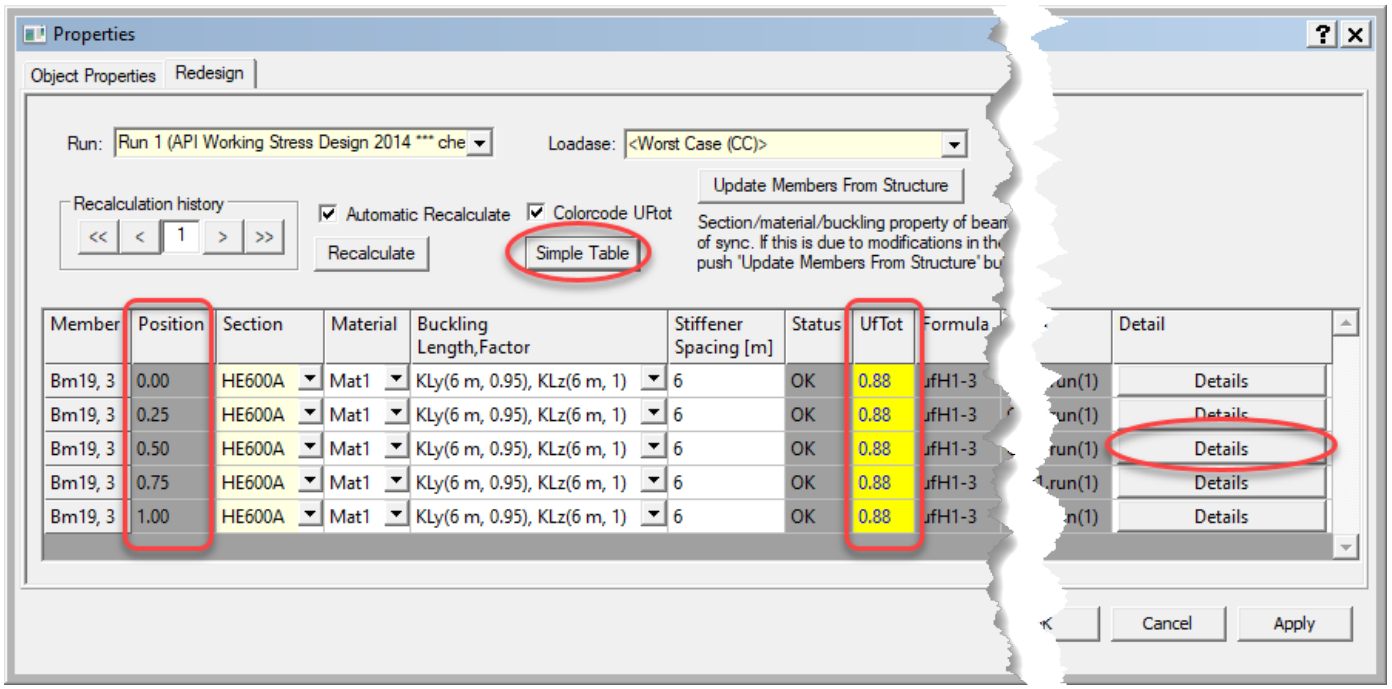


- See that the reduction of *UfTot* is immediate.



- As informed by the text in red, “Pressing OK/Apply will modify capacity model for current run”, clicking *OK* in the dialog effectuates the change of section and any other modification (*Material* and *Buckling Length, Factor*) done to the capacity member.

- To see more details, push the *Full Table* button (that below has turned into *Simple Table* to allow returning to one line per capacity member). All code check positions along the member with their usage factors *UfTot* are listed. (In this case all usage factors are equal because the member fails due to buckling that does not relate to any particular position along the member.)



- Click *Details* for a selected position to see all possible details. The data are tab separated to allow copy-pasting into e.g. Excel as done below.

Code check details for Bm19, 3	
ufShear	0.00728286
ufEuler	0
ufH1-1	0
ufH1-1ax	0
ufH1-1mo	0
ufH1-2	0
ufH1-2ax	0
ufH1-2mo	0
ufH1-3	0.879443
ufH1-3ax	0.00171409
ufH1-3mo	0.877728
ufH2-1	0
ufH2-1ax	0
ufH2-1mo	0
sldTens	84.9657
sldComp	84.9657
stfReqLimit	0
PGwebSldLimit	0
FyLim65ksi	0
OutOfBounds	0
relpos	0.5
Fy	360000

relpos	0.5
Fy	360000
E	2.10E+08
KLy	5.7
KLz	6
a	6
L	6
Lc	3.23366
Lb	6
h/tw	41.5385
b/tf	6
Cmy	1
Cmz	1
P	-5.03491
Vy	7.55941
Vz	-0.15532
My	21.4435
Mz	90.5458
Mymax	-27.8219
Mzmax	-171.575
FlaClass	0
WebClass	0
FlaAsWeb	0
WebAsFla	0

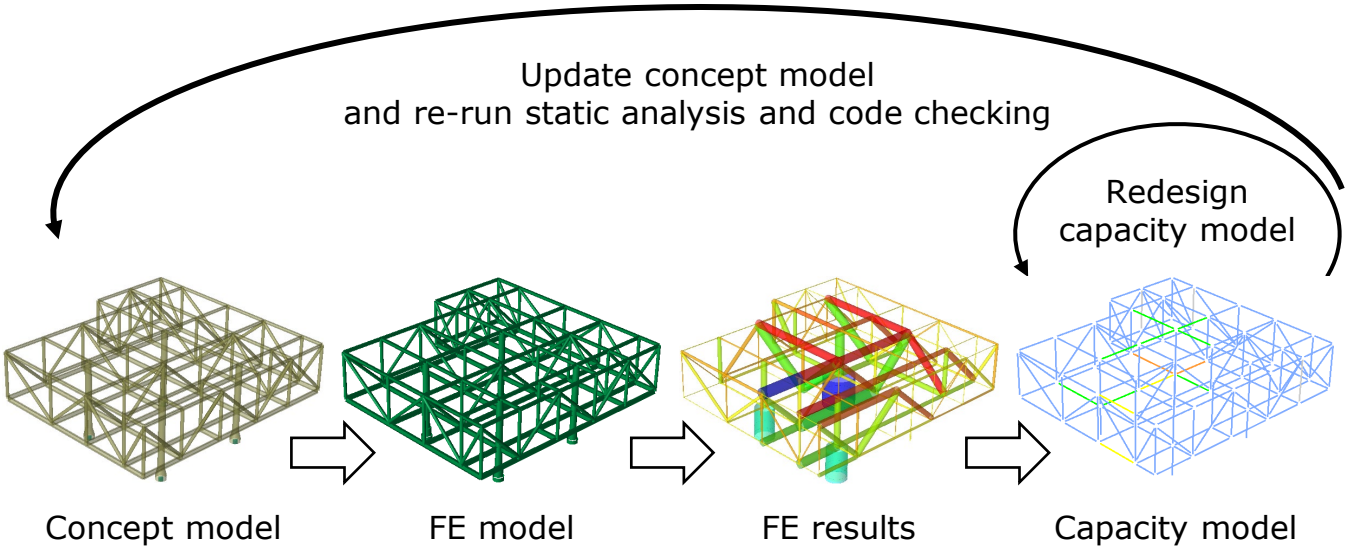
WebAsFla	0
Cb	1
Cv	0
fa	-222.784
fvy	755.941
fvz	-20.2504
fby	-5812.66
fbz	-228462
Qs	1
Qa	1
Fa	129972
Ft	216000
Fvy	144000
Fvz	144000
Fby	184103
Fbz	270000
Fey	2.08E+06
Fez	149791
J	4.07E-06
Cw	8.99E-06
fnw	0
fvw	0

➢ **Redesign Loop**

- The approach above (Modify Capacity Member Approach) is in short:

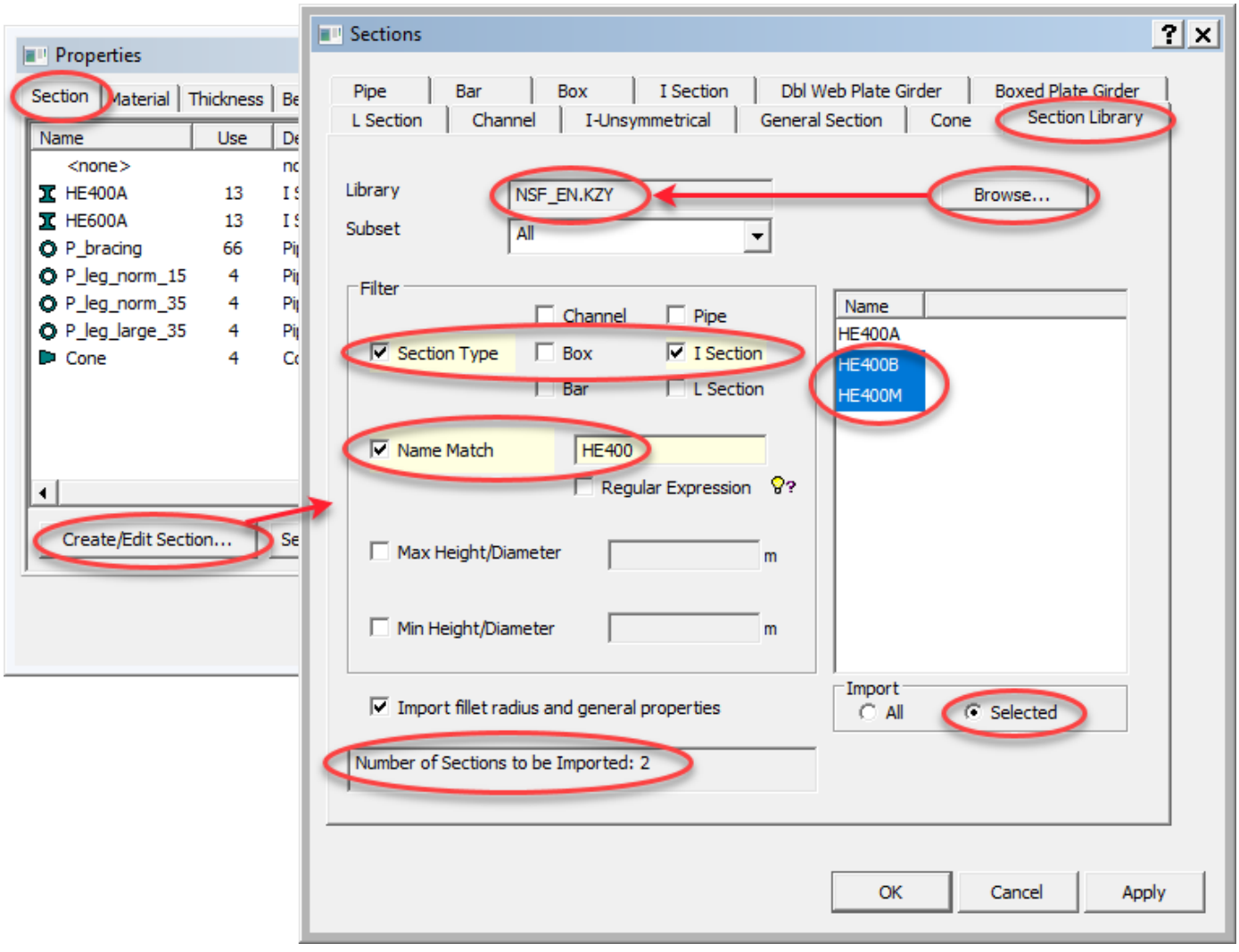
1. Redesign *capacity model* by modifying cross section and/or material.
   - Gives approximate usage factors based on constant member forces.
   - If all usage factors OK then go to step 2.
   - If not OK then go to step 1.
2. Update the *concept model* and rerun static analysis and code checking.
   - Creates new FE model and new FE results, i.e. new member forces.
   - If all usage factors OK then done.
   - If not OK then go to step 1.

- The Modify Capacity Member Approach presented graphically:

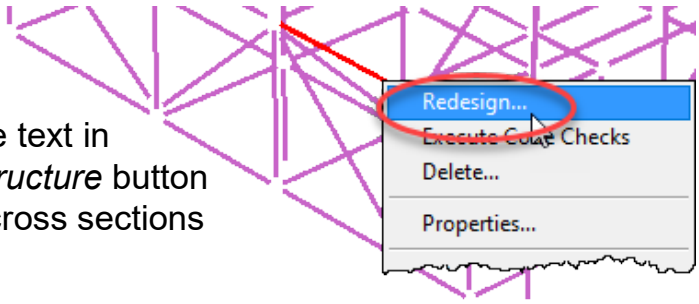


➢ Create/Import New Cross Sections

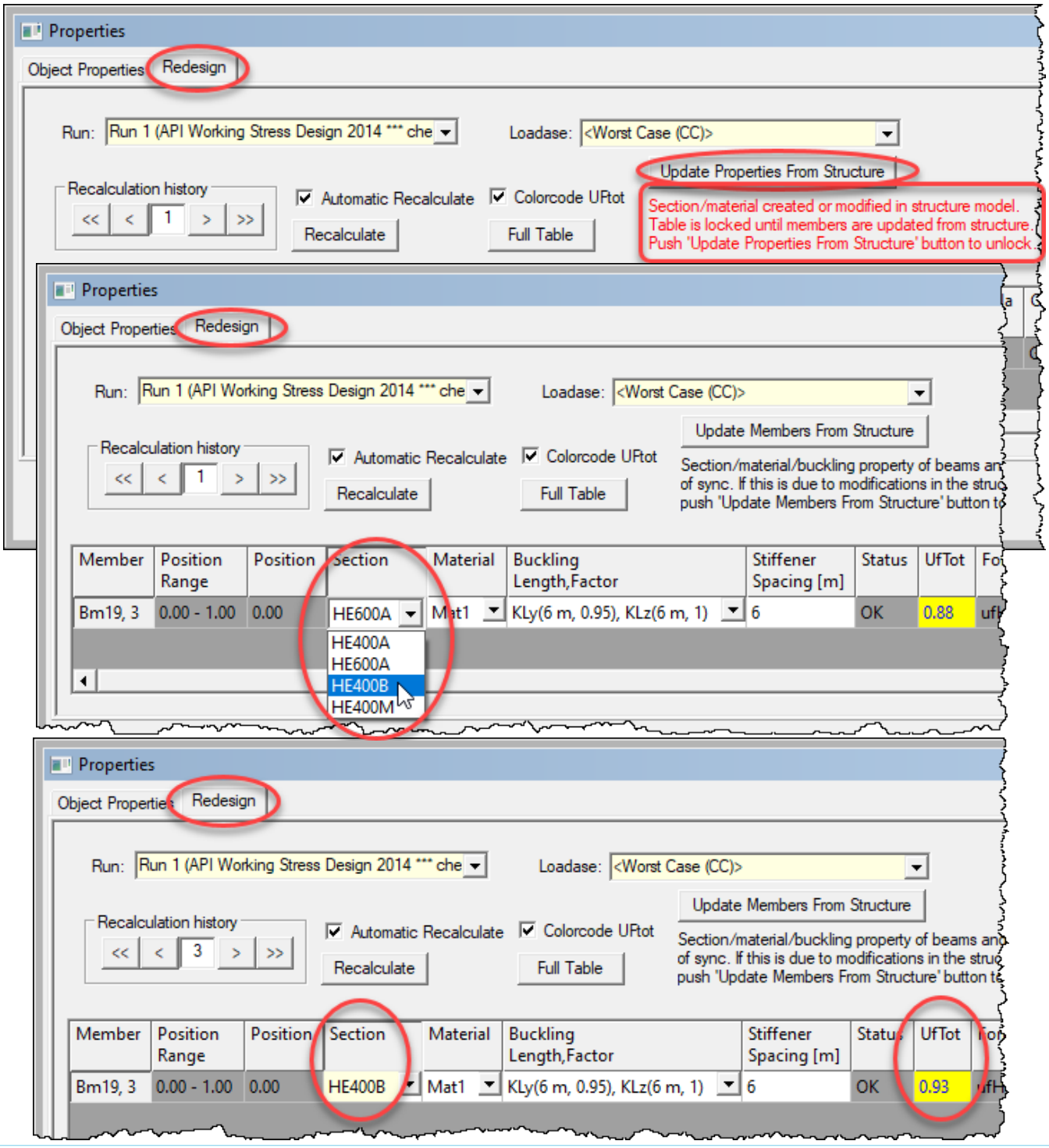
- In the redesign of the capacity model above, the cross section HE600A is assigned to a beam at the edge of the main deck. But since the cross section HE400A is the one used for other beams in the main deck, the higher cross section HE600A may be undesirable.
- Therefore, import more beam cross sections from the cross section library NSF_EN.KZY.
  - Use *Edit* | *Properties* to open the *Properties* dialog and go to the *Section* tab. Click the *Create/Edit Section* button.
  - In the *Sections* dialog go to the *Section Library* tab and browse to find the library NSF_EN.KZY. Filter the list of sections as shown and select the sections HE400B and HE400M. Click *OK* to import these sections.



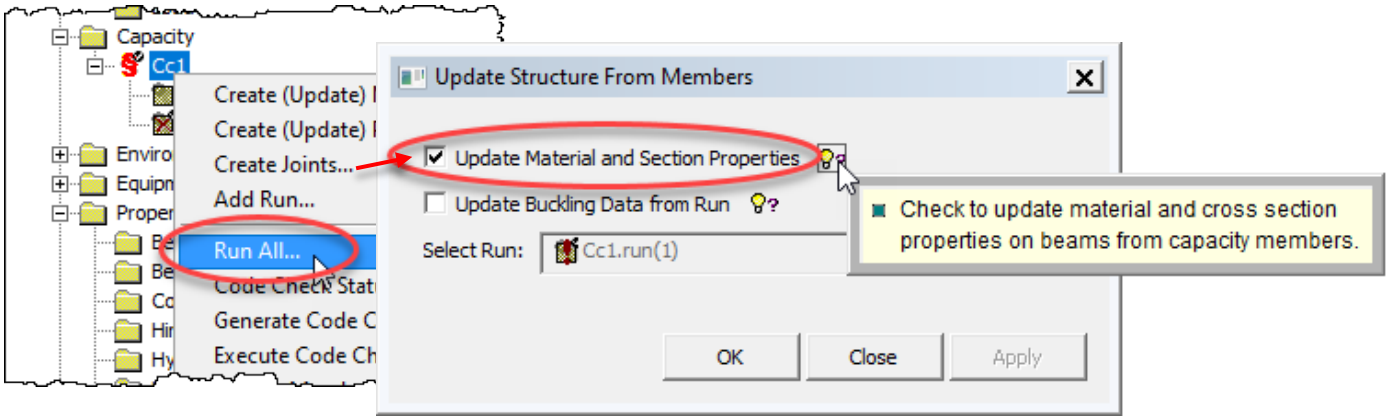
- Select the failing beam anew and right-click to select *Redesign*.



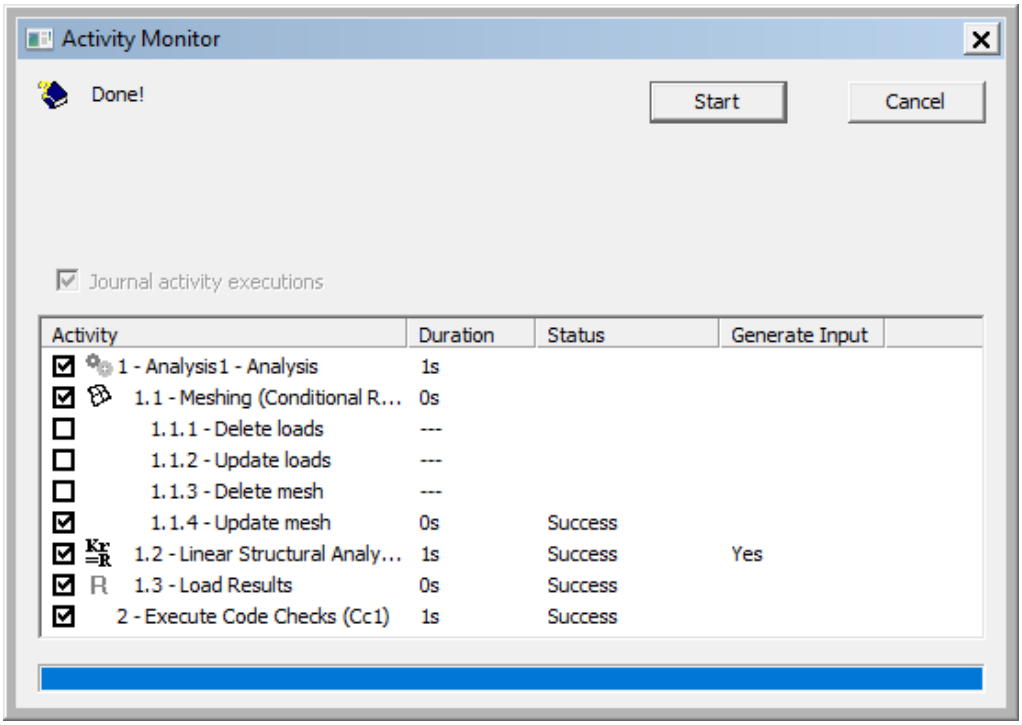
  - At this point, and as informed by the text in red, the *Update Properties From Structure* button must be pressed to make the new cross sections accessible to the capacity model.
  - One of the new sections may then be selected in the redesign.



- Again, the redesign has been done without rerunning the static analysis. I.e. the redesign is based on the assumption that the increase of section from HE400A to HE400B (with greater flange and web thicknesses) will not change the forces and moments in the redesigned beam. This of course not the case. The stronger cross section will involve a redistribution of loads in the deck structure and cause somewhat larger forces and moments in the redesigned beam.
- Again, to account for this, right-click the capacity manager in the browser, right-click and select *Run All*.
  - This time checking *Update Material and Section Properties* is required. Also see the help text appearing when hovering the mouse over the light bulb symbol.

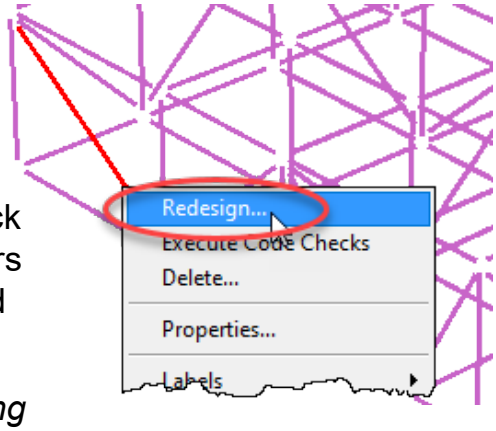


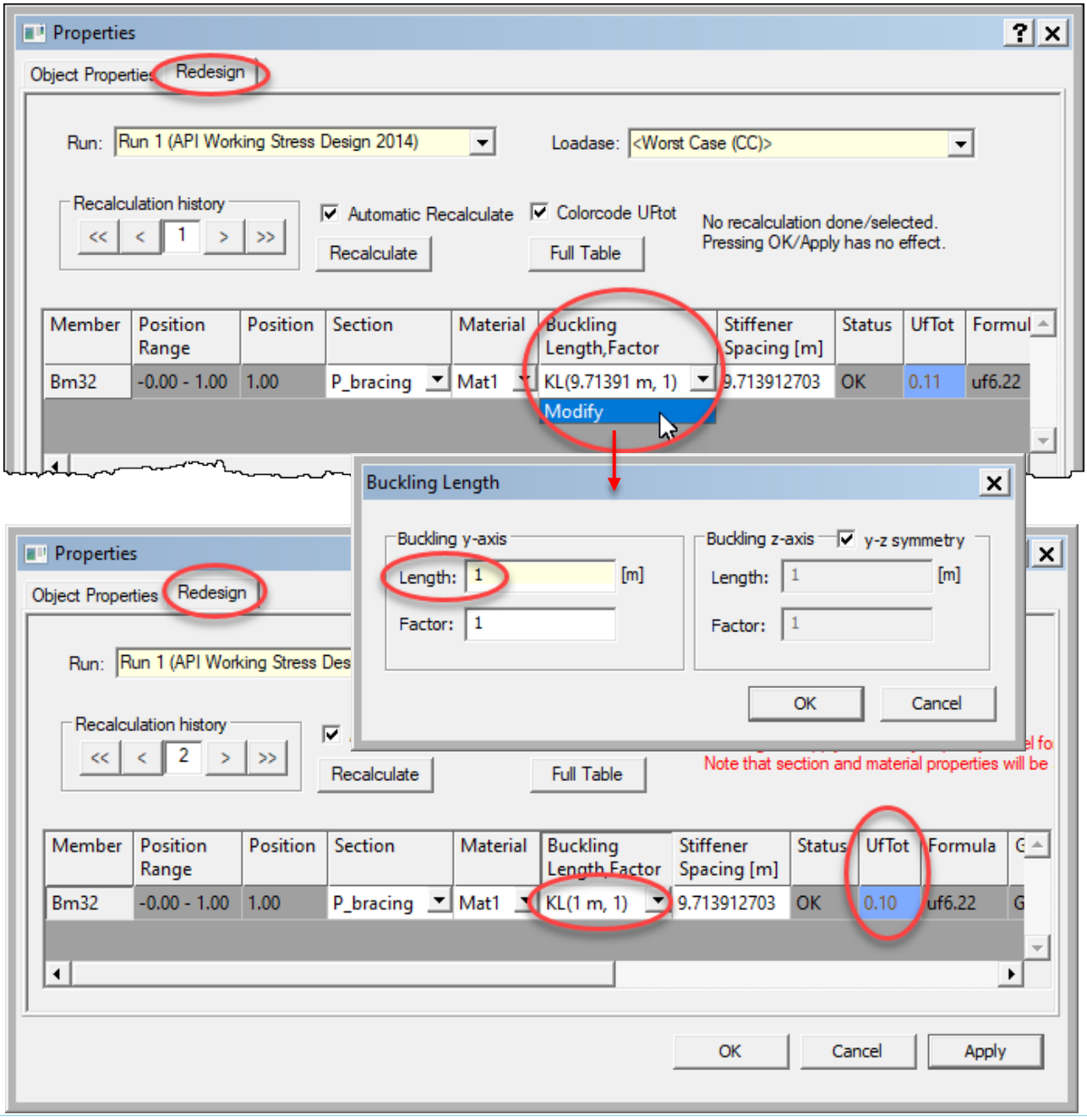
  - Click *Start* to rerun the static analysis as well as executing the code check. This is often referred to as a design iteration.



➢ Modify Buckling Data

- The redesign also allows modifying buckling data. Modifying buckling data does not alter the model stiffness so rerunning the analysis is unnecessary.
- Select a beam with compressive force and right-click to select *Redesign*. Our model contains no members for which buckling length is of concern, the selected one still demonstrates the process.
  - In the pull down menu click *Modify*. In the *Buckling Length* dialog give e.g. *Length*=1. The *UfTot* changes slightly.

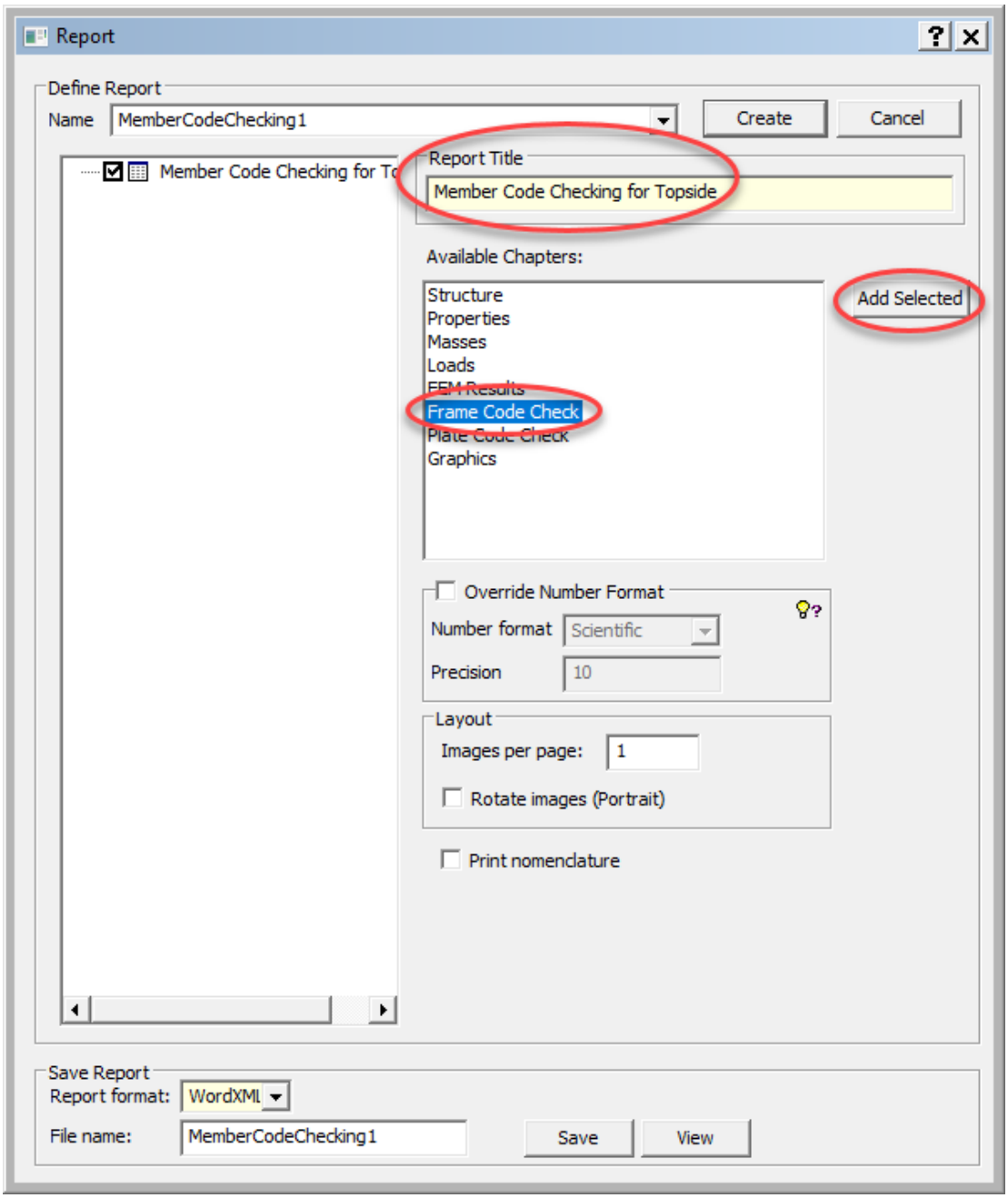




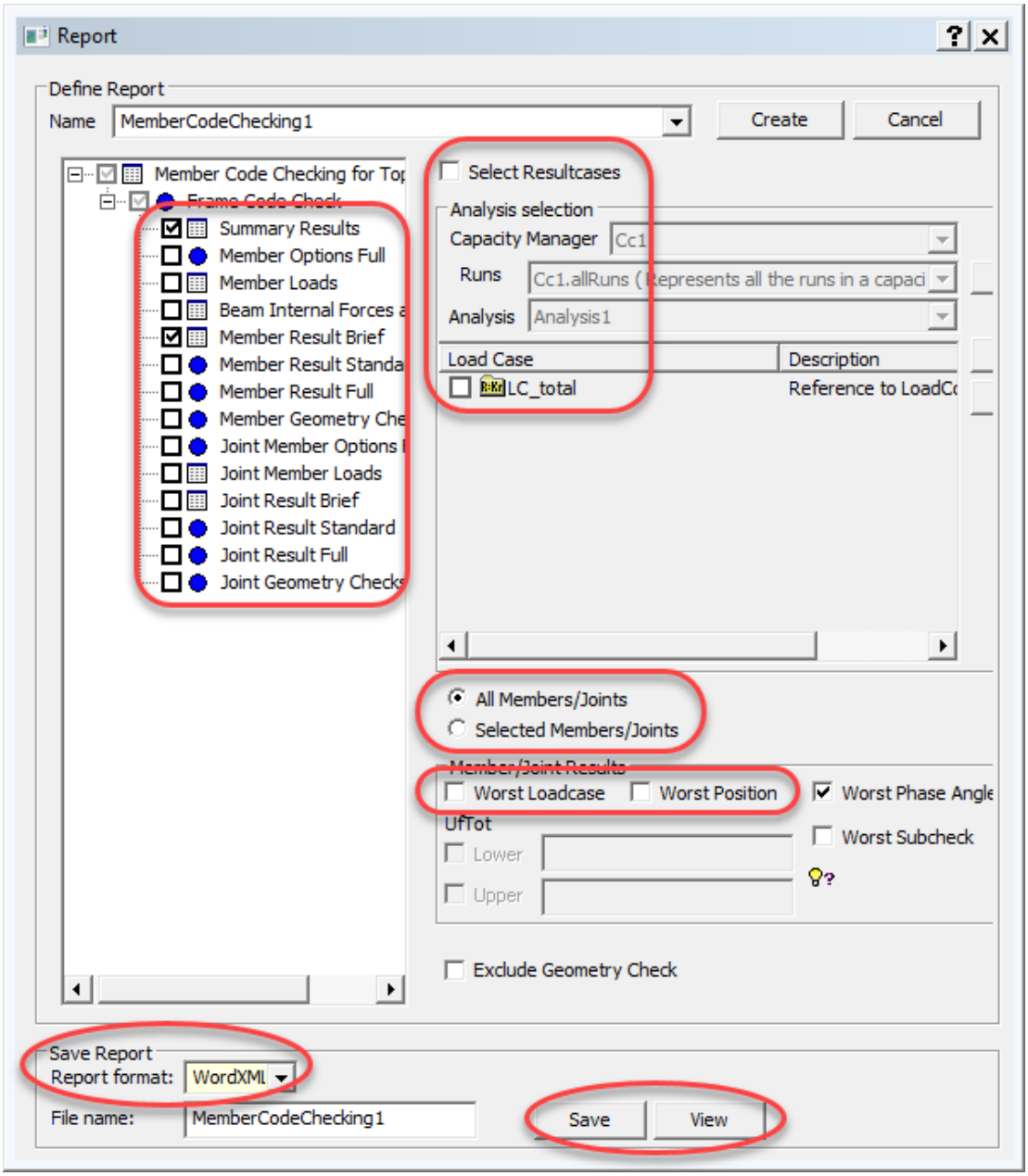
9 CREATE CODE CHECK REPORT

➢ Use *File* | *Save report* to create a report with code check results.

- The report *Name* is the object name in the GeniE database and is not exposed in the report.
- The *Report Title* appears in the report, adjust it to your liking.
- Select among the *Available Chapters*, e.g. limit the report to a chapter on *Frame Code Check*, and click *Add Selected*.



- The field to the left in the *Report* dialog now shows the contents of the report being created. Refine the contents by checking and unchecking subchapters.
- Check *Select Resultcases* unless all cases shall be reported.
- Decide whether to include *All Members* or only *Selected Members*.
- Decide whether to limit the report to *Worst Loadcase* and/or *Worst Position*.
- Select *Report format*, e.g. WordXML (other options are ExcelXML and html).
- Click *Save* and then *View*. If you have another application associated with the XML file extension, Word may not open. Then go to the workspace folder (*File* | *Explore Current Workspace*) and open the XML file in Word.



- The report opens in Word.
  - Right-click the table of contents and select *Update Fields* to update page numbers.

Table of Contents

  - An excerpt of the report is shown below.

1 Cc1 : Frame Code Check

Description : Capacity Manager

1.1 Cc1.run(1) : Frame Code Check

Description : API Working Stress Design 2014

General options

Code	API WSD 2014
CapendIncluded	true
Azimuthal Tolerance Angle	5
Ind. Brace Can Distance	true
Joint Geometric Limits	Not checked
Minimum Cut Off Value	0
Brace Unit Check Percentage	85 %
Use 50% Effective Strength Check	false
Check Minimum Required Thickness	false
Incremental Thickness Value	0.001
Stop criteria	Radius

General options

Code	AISC 9TH
ReferenceEmodKSI	29000
Purge position results	false

1.1.1 Cc1.run(1) : Summary Results

Cc1.run(1) : Summary Results

- Sorted by Set (Ascending)

- Run : Cc1.run(1)
- Worst LoadCase per Capacity Model (Member or Joint)
- Worst SubCheck per Capacity Model (Member or Joint)
- Worst Position along Member / Worst Brace Member of Joint
- Worst 3 Members / Joints per Set

Set	Count	UfTot > 1.33	UfTot > 1.00	UfTot > 0.80	UfTot > 0.50	UfTot > 0.01	Below	CapModel	LoadCase	Position	Status	UfTot	Run
Total	180	0	3	2	9	163	3	Bm19, 3	LC_total	0.00	Failed(uf)	1.17	Cc1.run(1)
Total								Bm4, 2	LC_total	0.00	Failed(uf)	1.16	Cc1.run(1)
Total								Bm11, 4	LC_total	0.00	Failed(uf)	1.08	Cc1.run(1)
Cellar_deck	53	0	2	2	3	46	0	Bm4, 2	LC_total	0.00	Failed(uf)	1.16	Cc1.run(1)
Cellar_deck								Bm11, 4	LC_total	0.00	Failed(uf)	1.08	Cc1.run(1)
Cellar_deck								Bm11, 3	LC_total	0.00	OK	0.90	Cc1.run(1)
Main_deck	53	0	1	0	6	46	0	Bm19, 3	LC_total	0.00	Failed(uf)	1.17	Cc1.run(1)
Main_deck								Bm24, 3	LC_total	1.00	OK	0.72	Cc1.run(1)
Main_deck								Bm24, 4	LC_total	0.00	OK	0.71	Cc1.run(1)

About DNV

We are the independent expert in risk management and quality assurance. Driven by our purpose, to safeguard life, property and the environment, we empower our customers and their stakeholders with facts and reliable insights so that critical decisions can be made with confidence. As a trusted voice for many of the world's most successful organizations, we use our knowledge to advance safety and performance, set industry benchmarks, and inspire and invent solutions to tackle global transformations.

Digital Solutions

DNV is a world-leading provider of digital solutions and software applications with focus on the energy, maritime and healthcare markets. Our solutions are used worldwide to manage risk and performance for wind turbines, electric grids, pipelines, processing plants, offshore structures, ships, and more. Supported by our domain knowledge and Veracity assurance platform, we enable companies to digitize and manage business critical activities in a sustainable, cost-efficient, safe and secure way.

DNV | digital@dnv.com | dnv.com/digital