SESAM TUTORIAL

GeniE

Jacket Member and Joint Code Checking

Valid from program version 8.2

Sesam Tutorial

GeniE – Jacket Member and Joint Code Checking

Date: June 2021

Valid from GeniE version 8.2

Prepared by: Digital Solutions at DNV

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

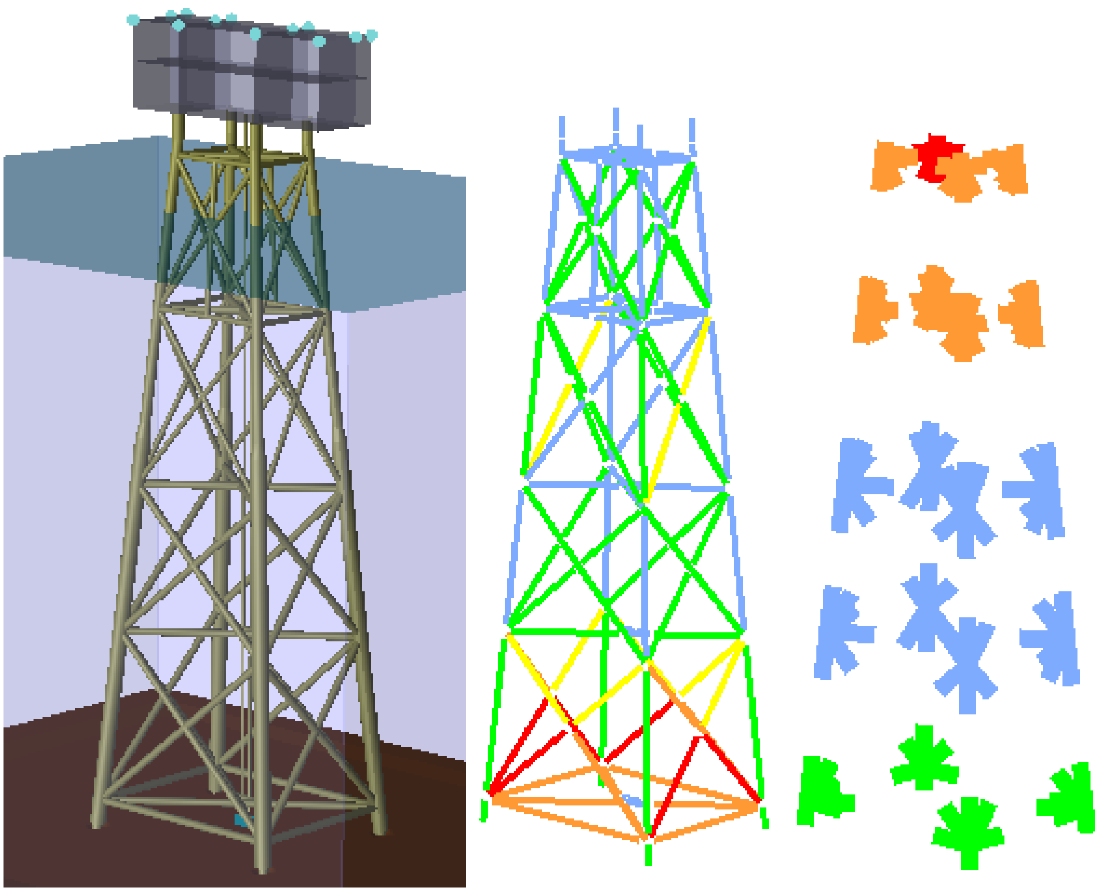
E-mail sales: digital@dnv.com

DNV

TABLE OF CONTENTS

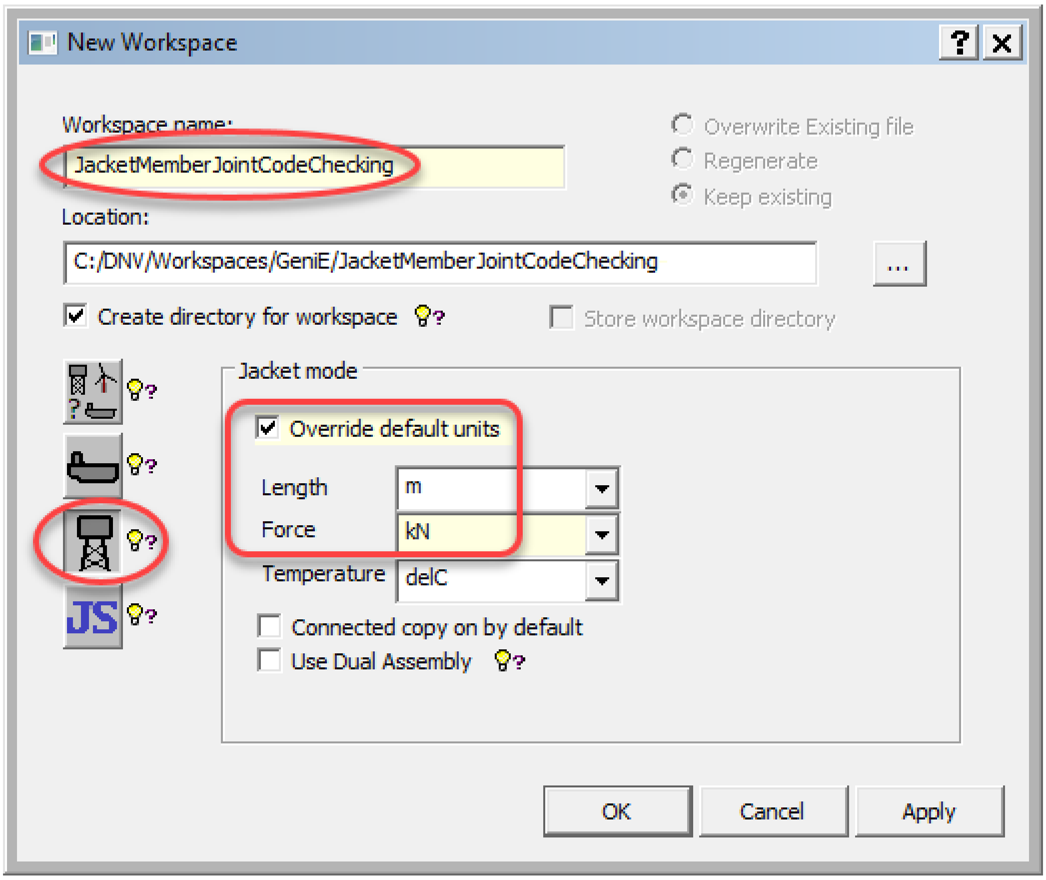
1 INTRODUCTION

- In this tutorial member and joint code checking according to API WSD is performed for the jacket created in tutorial 'GeniE Piled Jacket Analysis'.
- This tutorial contains the following steps:
  - Open a new workspace and read a js file creating and analysing the jacket.
  - Create joint concepts.
  - Create a capacity manager with capacity members, joints and code check run.
  - Generate code check loads and execute code checking.
  - Investigate and redesign a failing member and joint.
  - Rerun structural analysis and code checking after redesign.
- Go to the tutorial 'GeniE Member Code Checking' to learn more about beam forces and moments, the redesign loop and creating code check reports.
- To complete this tutorial you need license to GeniE including code checking, Wajac (wave load calculation), Splice (pile-soil analysis) and Sestra (structural analysis).
- The jacket model plus member and joint capacity models are shown below.



2 OPEN NEW WORKSPACE AND IMPORT MODEL

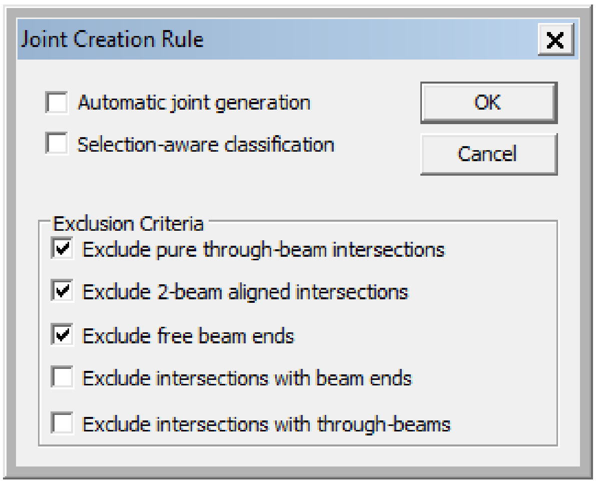
- Start GeniE and open a new workspace.
  - Give a *Workspace* name.
  - Click the *Jacket mode* button to customise for jacket modelling, i.e. limit menus and buttons to those relevant for jacket modelling.
  - Set units m and kN and click *OK*.

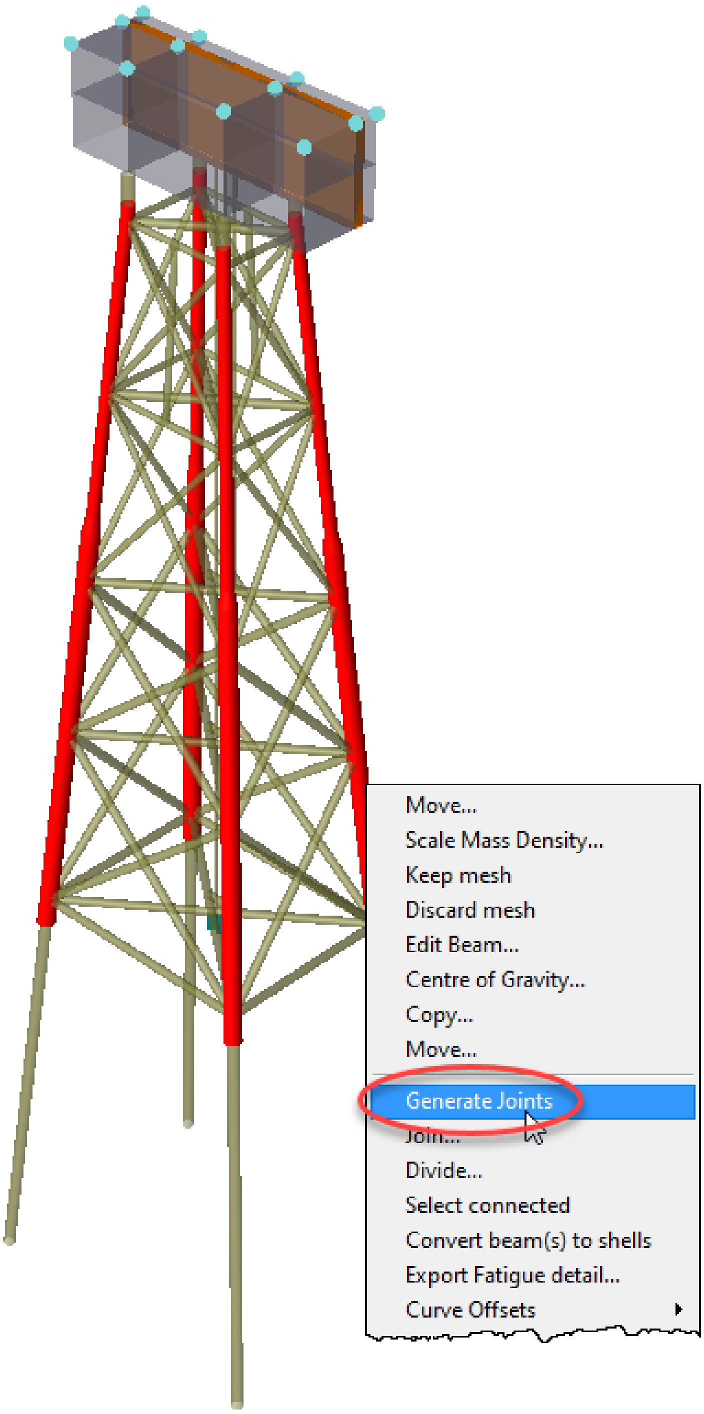


  - If the workspace name exists, select *Overwrite Existing file* or give another name.
- Use *File* | *Read Command File* to read the js file of tutorial ‘B8 GeniE Piled Jacket Analysis’ named PiledJacketAnalysis_input.js. Tutorials are found in the installation folder typically named <path>\GeniE VX.Y-ZZ\Help\Tutorials\.
  - See that a static analysis is run as the last step of this js file.
  - Open the *Analysis* | *Activities* | *WaveLoadAnalysis* folder and see that the analysis contains eight load combinations.
  - All combinations have been assigned operating design condition. This can be seen by right-clicking a combination, selecting *Properties* and going to the *Design Condition* tab.

3 CREATE JOINT CONCEPTS

- The imported model does not contain any joint concepts.
  - Joint concepts are used for designing tubular joints with gaps, cans, stubs and cones but are not a requirement for doing a structural analysis. This means that no joint design has been done for the imported model. From this follows that the model may have joints with overlaps which is in general bad design.
  - Joints are, however, required for joint code checking. Joints must, therefore, be created.
  - Select the four legs. This can e.g. be done by selecting the set named Legs found in the *Utilities* | *Sets* | *Regular Sets* folder.
  - Right-click the legs and click *Generate Joints* to create joints along the legs.
    - By *Edit* | *Rules* | *Joint creation* control can be exerted over where to generate joints. In this case the default options are OK.

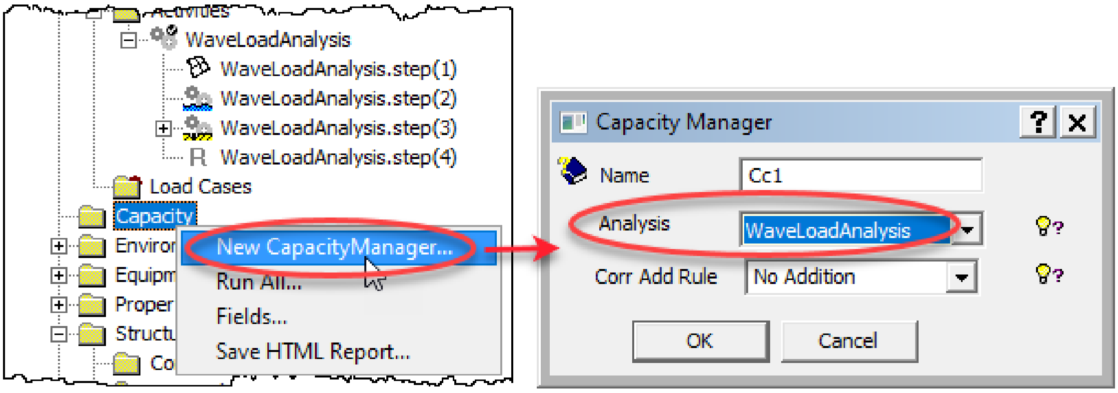




  - To see the joints, right-click the *Joint selection* button ( ) and open 'the eye'. Joints are displayed as brown balls.
  - To enable selecting joints the *Joint selection* button must be depressed.
  - The joints are found in the *Structure* folder where they also can be selected.

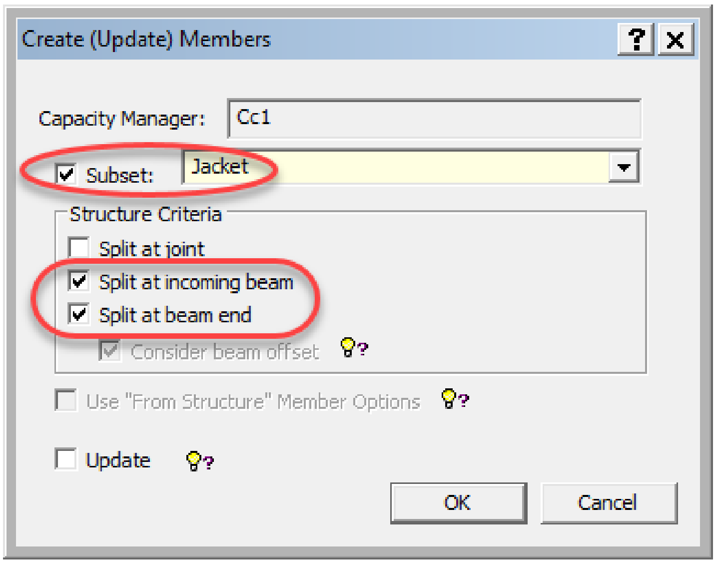
4 CREATE CAPACITY MANAGER AND FILL WITH CAPACITY MEMBERS

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

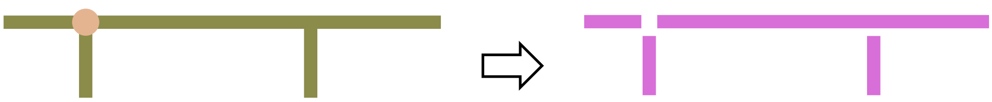


➢ Right-click the capacity manager and select *Create (Update) Members* to fill it with capacity members.

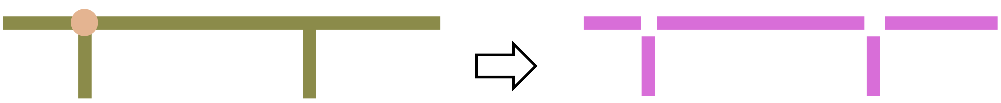
- In the *Create (Update) Members* dialog check *Subset* and select the set named Jacket for code checking. I.e. do not code check the complete model.



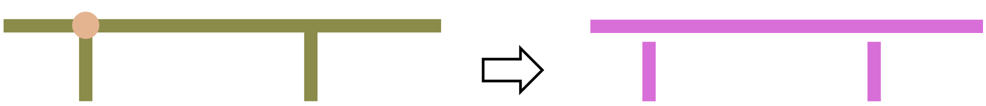
- Also select the options *Split at incoming beam* and *Split at beam end*. The three options explained:
  - *Split at joint* involves splitting at joints when creating members from beams.



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



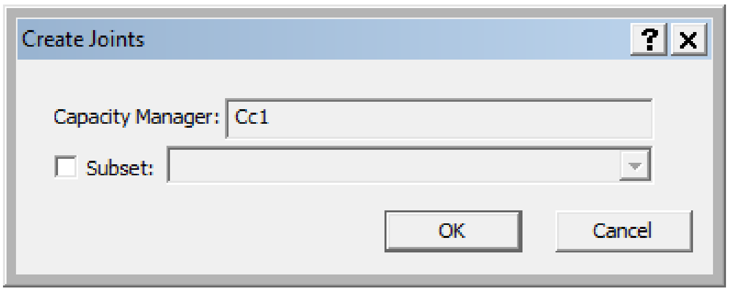
  - *Split at beam end* involves no splitting of beams when creating members.



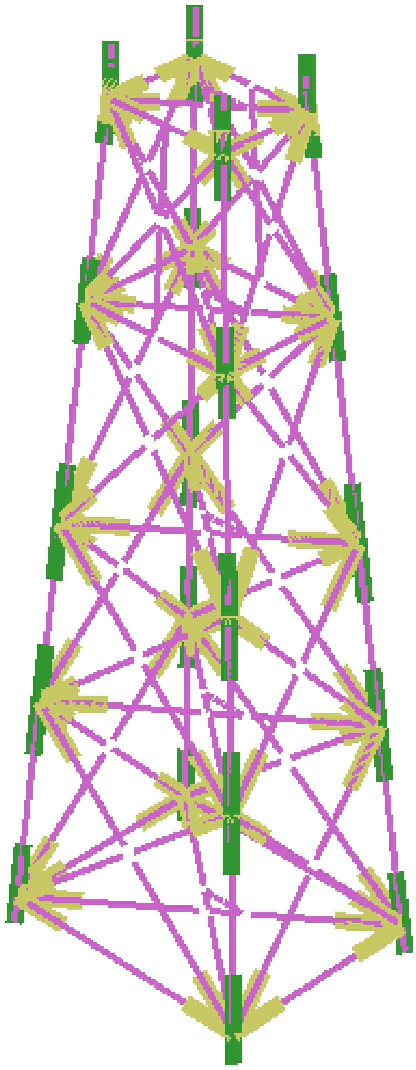
➢ Switch to *Capacity Models* display configuration to see the capacity members.

5 FILL CAPACITY MANGER WITH CAPACITY JOINTS

- Right-click the capacity manager and select *Create Joints* to fill it with capacity joints.
  - Alternatively, a second capacity manager may be created and used for capacity joints so as to split capacity joints from capacity members.

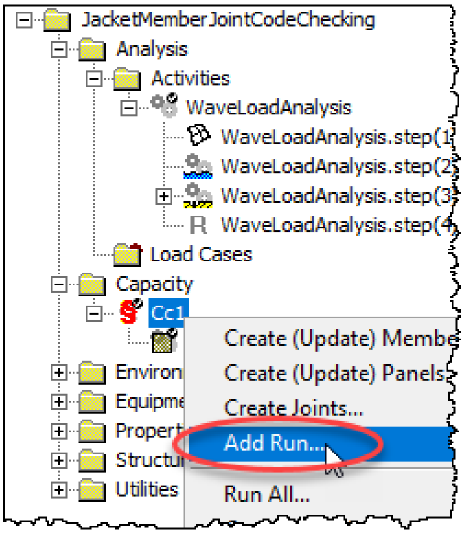


- The *Capacity Models* display configuration now shows the capacity members in pink and capacity joints in green and yellow as shown below in wireframe mode.

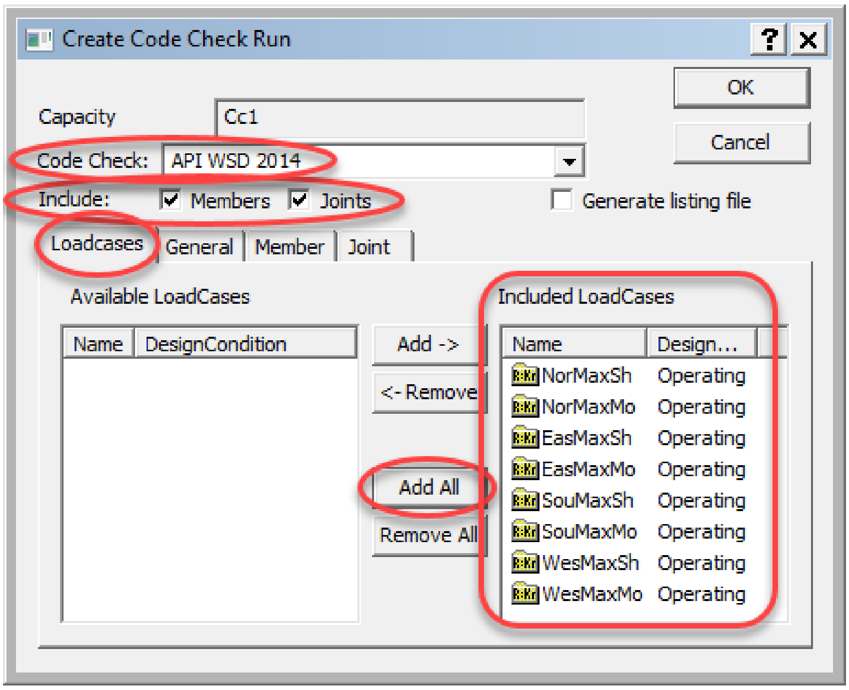


6 ADD CODE CHECK RUN

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

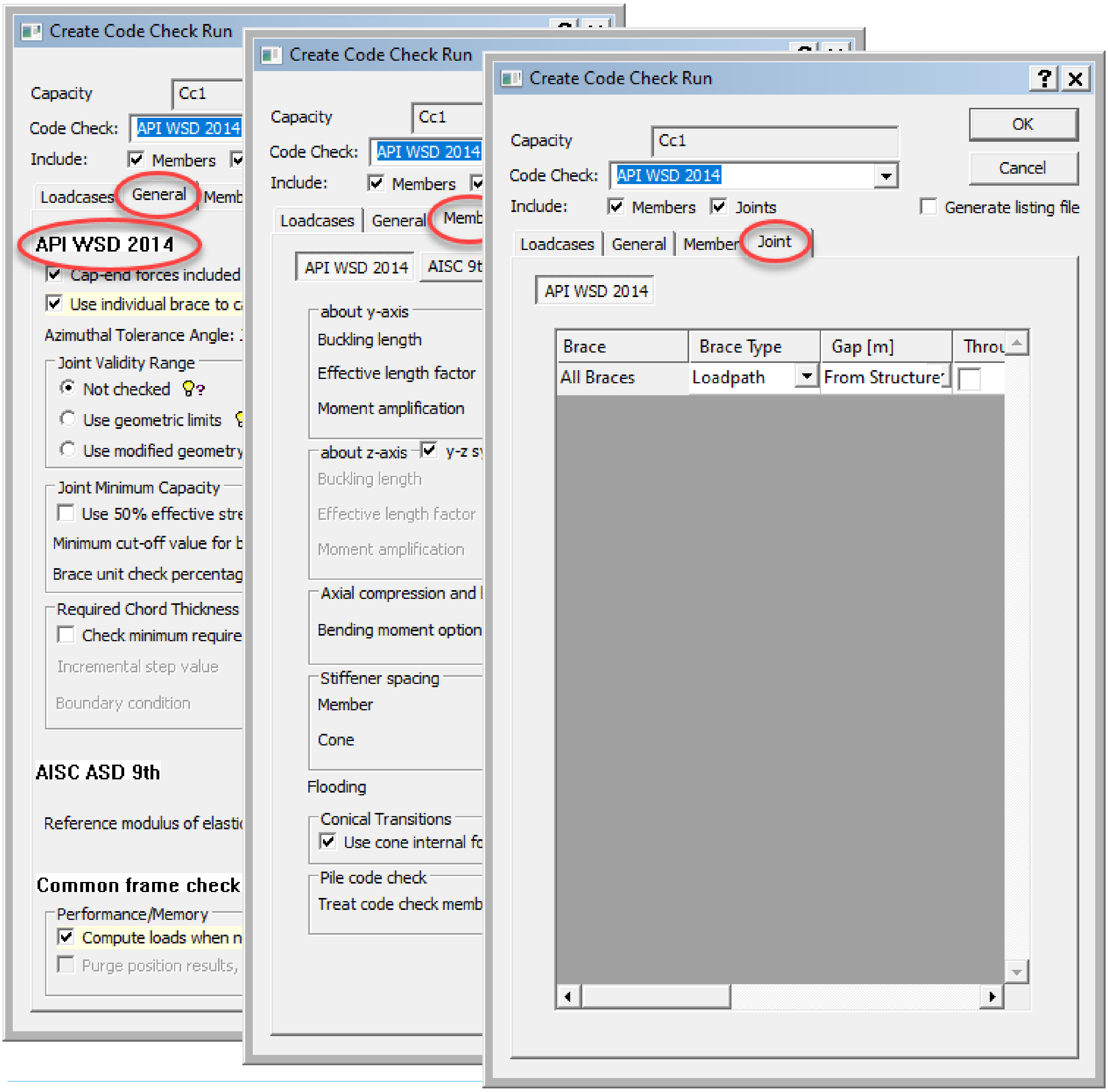


- In the dialog select *Code Check API WSD 2014*.
- Check both *Members* and *Joints* as both members and tubular joints shall be code checked.
  - Alternatively, check only *Members* and create a second run for joints.
- In the *Loadcases* tab, click *Add All* to move all combinations to the *Included LoadCases* field. I.e. code check all combinations.



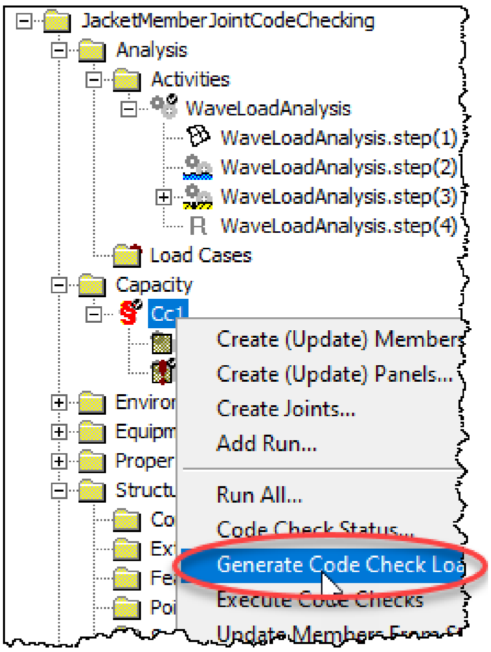
- Don't *OK* the dialog yet as the next page deals with the *General*, *Member* and *Joint* tabs of the *Create Code Check Run* dialog.

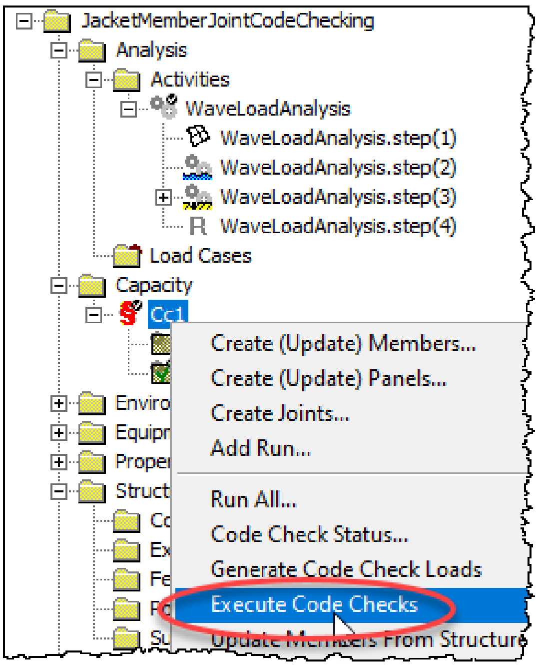
- In the *General* tab some general options may be modified. Notice that the dialog informs that while tubular members are checked according to *API WSD 2014*, non-tubulars, if any, are checked according to *AISC ASD 9th*.
- In the *Member* tab *Buckling length*, *Effective length factor*, *Moment amplification* 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.
- In the *Joint* tab *Brace Type*, *Gap* and other joint checking data are specified for *all* joints. These data are modified for selected joints as explained later.
- In this tutorial, accept all default settings in these three tabs.

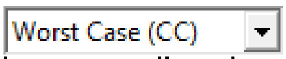


7 GENERATE CODE CHECK LOADS AND EXECUTE CODE CHECKS

- Right-click the code check run and click *Generate Code Check Loads*. For a big model with many load cases/combinations selected for code checking this may require a few seconds.
  - At this point a dialog may appear saying that the concept model may be more recent than the analysis. This is because joint concepts were added after running the analysis. As joint concepts have no consequence for the analysis answer yes to proceed.
  - 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.



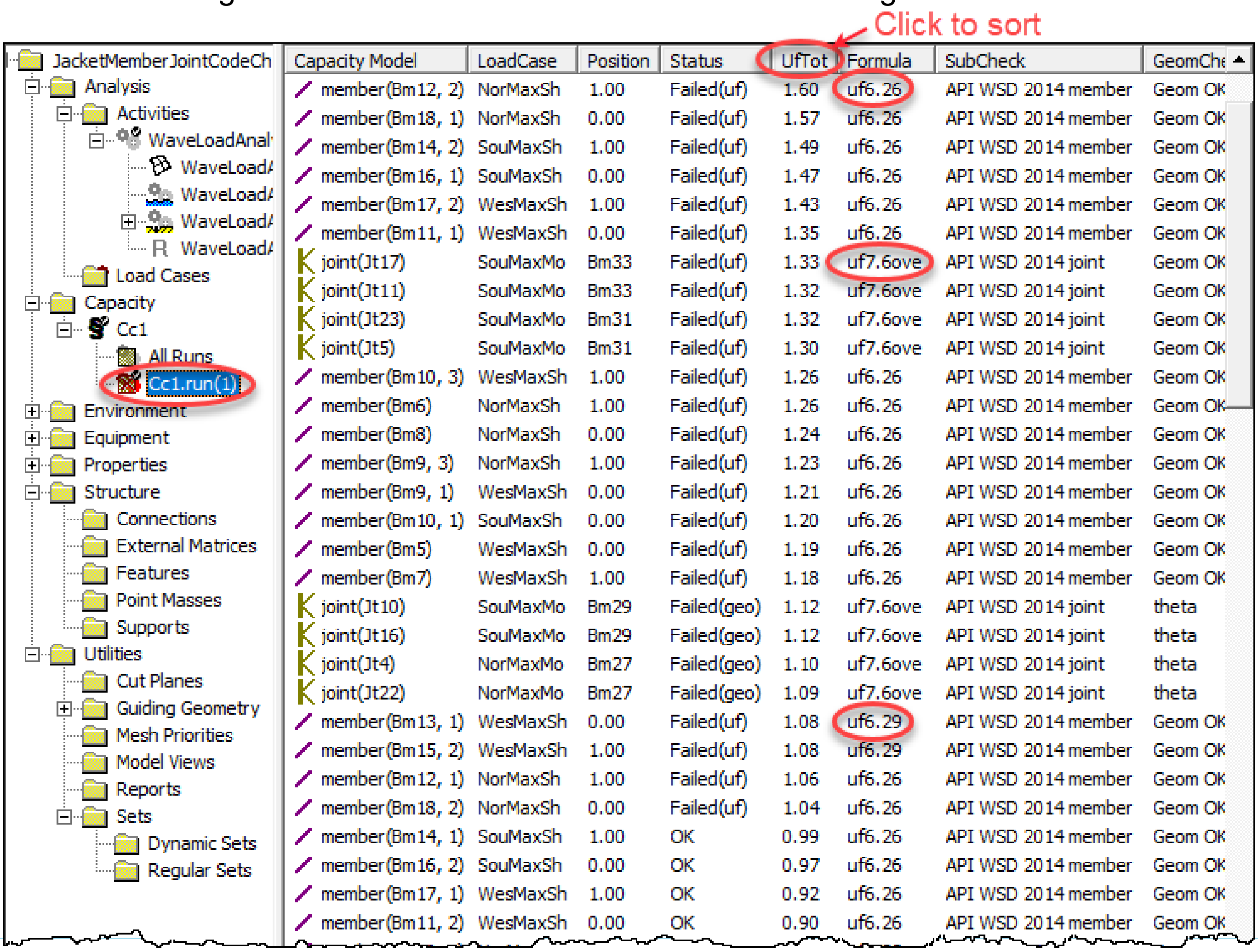


- In the load case selector a case named *Worst Case (CC)* appears: 

Select this which is an envelope case containing the highest utilisation over all code checked load combinations.

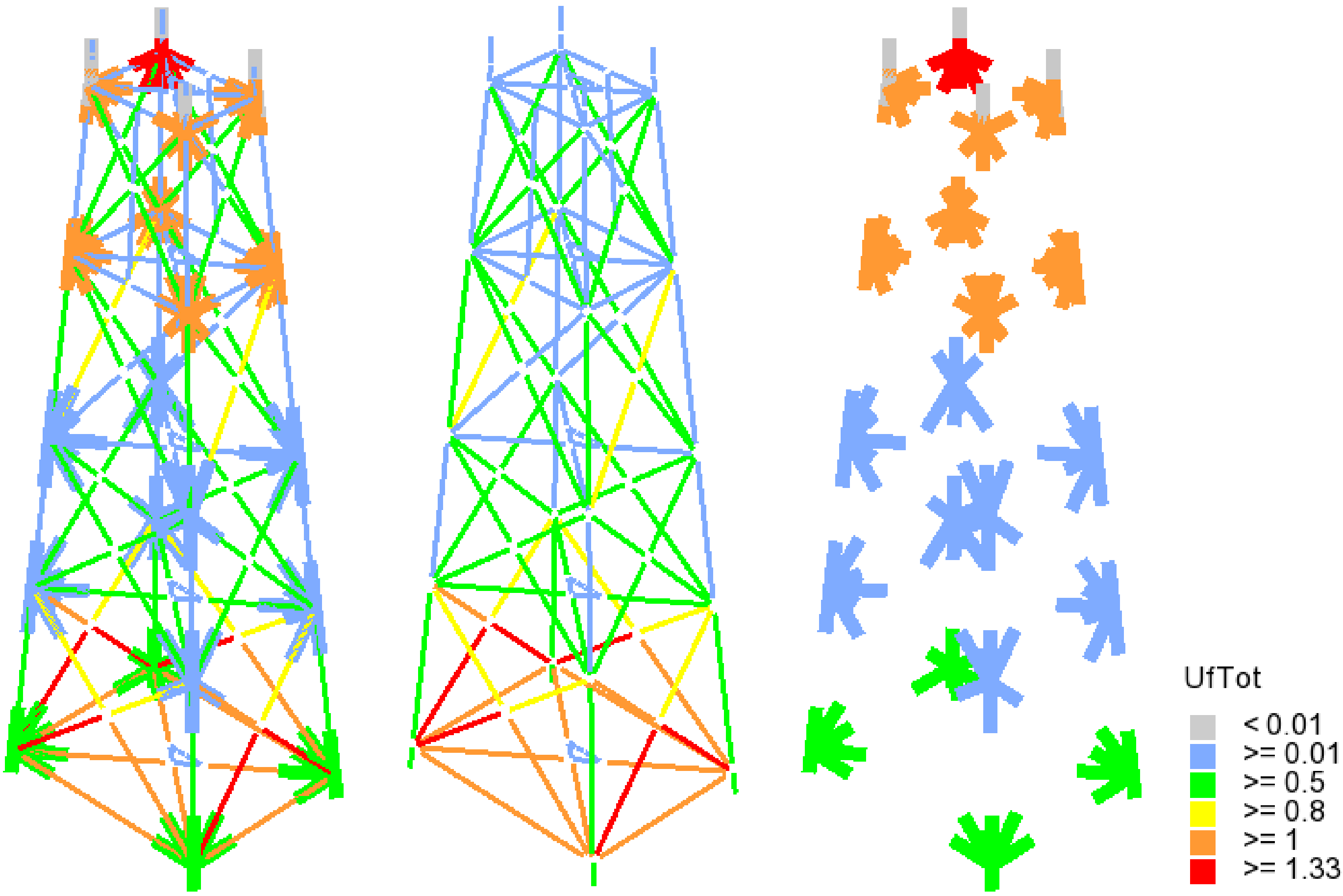
➢ The browser now tabulates the code check results. For each capacity member (the name refers to beam and capacity member number within it), the most severe load case/combination and position along it [0, 1] is listed. For each capacity joint the *Position* column identifies the critical brace. *Status* = *Failed(uf)* means failure and the usage factor is given in the next column *UfTot*, the largest 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:

- *uf6.26* means 'Usage factor according to equation (6.26), i.e. axial tension and hydrostatic pressure'.
- *uf7.6ove* means 'Usage factor from overlap brace in overlapping joint, through brace as chord'.
- *uf6.13* means 'Usage factor according to equation (6.13), i.e. hydrostatic pressure'.
- Note that four joints are listed without any results (excluded from the table below). These are the 'joints' at the transition from the inclined legs to the vertical stubs. These might have been excluded from the code checking.

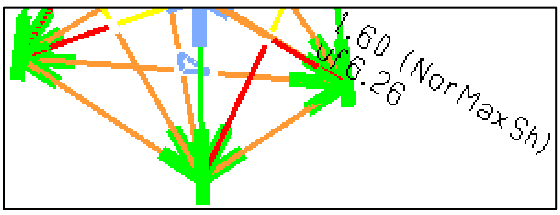


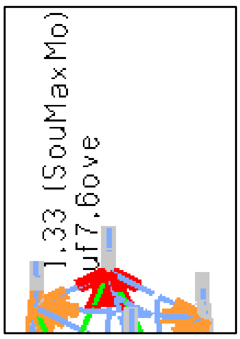
Capacity Model	LoadCase	Position	Status	UfTot	Formula	SubCheck	GeomChe
member(Bm12, 2)	NorMaxSh	1.00	Failed(uf)	1.60	uf6.26	API WSD 2014 member	Geom OK
member(Bm18, 1)	NorMaxSh	0.00	Failed(uf)	1.57	uf6.26	API WSD 2014 member	Geom OK
member(Bm14, 2)	SouMaxSh	1.00	Failed(uf)	1.49	uf6.26	API WSD 2014 member	Geom OK
member(Bm16, 1)	SouMaxSh	0.00	Failed(uf)	1.47	uf6.26	API WSD 2014 member	Geom OK
member(Bm17, 2)	WesMaxSh	1.00	Failed(uf)	1.43	uf6.26	API WSD 2014 member	Geom OK
member(Bm11, 1)	WesMaxSh	0.00	Failed(uf)	1.35	uf6.26	API WSD 2014 member	Geom OK
joint(Jt17)	SouMaxMo	Bm33	Failed(uf)	1.33	uf7.6ove	API WSD 2014 joint	Geom OK
joint(Jt11)	SouMaxMo	Bm33	Failed(uf)	1.32	uf7.6ove	API WSD 2014 joint	Geom OK
joint(Jt23)	SouMaxMo	Bm31	Failed(uf)	1.32	uf7.6ove	API WSD 2014 joint	Geom OK
joint(Jt5)	SouMaxMo	Bm31	Failed(uf)	1.30	uf7.6ove	API WSD 2014 joint	Geom OK
member(Bm10, 3)	WesMaxSh	1.00	Failed(uf)	1.26	uf6.26	API WSD 2014 member	Geom OK
member(Bm6)	NorMaxSh	1.00	Failed(uf)	1.26	uf6.26	API WSD 2014 member	Geom OK
member(Bm8)	NorMaxSh	0.00	Failed(uf)	1.24	uf6.26	API WSD 2014 member	Geom OK
member(Bm9, 3)	NorMaxSh	1.00	Failed(uf)	1.23	uf6.26	API WSD 2014 member	Geom OK
member(Bm9, 1)	WesMaxSh	0.00	Failed(uf)	1.21	uf6.26	API WSD 2014 member	Geom OK
member(Bm10, 1)	SouMaxSh	0.00	Failed(uf)	1.20	uf6.26	API WSD 2014 member	Geom OK
member(Bm5)	WesMaxSh	0.00	Failed(uf)	1.19	uf6.26	API WSD 2014 member	Geom OK
member(Bm7)	WesMaxSh	1.00	Failed(uf)	1.18	uf6.26	API WSD 2014 member	Geom OK
joint(Jt10)	SouMaxMo	Bm29	Failed(geo)	1.12	uf7.6ove	API WSD 2014 joint	theta
joint(Jt16)	SouMaxMo	Bm29	Failed(geo)	1.12	uf7.6ove	API WSD 2014 joint	theta
joint(Jt4)	NorMaxMo	Bm27	Failed(geo)	1.10	uf7.6ove	API WSD 2014 joint	theta
joint(Jt22)	NorMaxMo	Bm27	Failed(geo)	1.09	uf7.6ove	API WSD 2014 joint	theta
member(Bm13, 1)	WesMaxSh	0.00	Failed(uf)	1.08	uf6.29	API WSD 2014 member	Geom OK
member(Bm15, 2)	WesMaxSh	1.00	Failed(uf)	1.08	uf6.29	API WSD 2014 member	Geom OK
member(Bm12, 1)	NorMaxSh	1.00	Failed(uf)	1.06	uf6.26	API WSD 2014 member	Geom OK
member(Bm18, 2)	NorMaxSh	0.00	Failed(uf)	1.04	uf6.26	API WSD 2014 member	Geom OK
member(Bm14, 1)	SouMaxSh	1.00	OK	0.99	uf6.26	API WSD 2014 member	Geom OK
member(Bm16, 2)	SouMaxSh	0.00	OK	0.97	uf6.26	API WSD 2014 member	Geom OK
member(Bm17, 1)	WesMaxSh	1.00	OK	0.92	uf6.26	API WSD 2014 member	Geom OK
member(Bm11, 2)	WesMaxSh	0.00	OK	0.90	uf6.26	API WSD 2014 member	Geom OK

- ➢ 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.
  - Capacity members and joints displayed together as shown to the left may be too congested. Split into capacity members and capacity joints by sorting the code check results table according to *Capacity Model*, selecting members only or joints only, and using Alt+S to display only the selection.



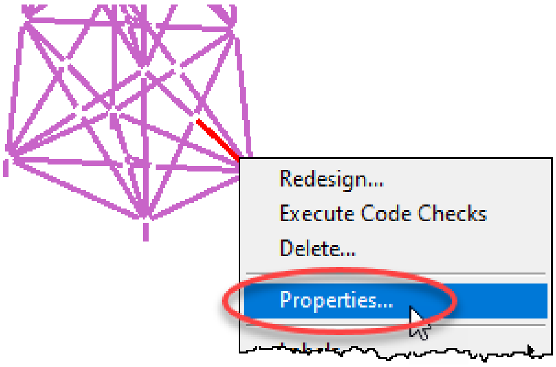
  - Note that you may right-click a capacity member/joint and select *Labels* | *Results* to label data like usage factor with governing load combination in parentheses, decisive code check equation and much more.



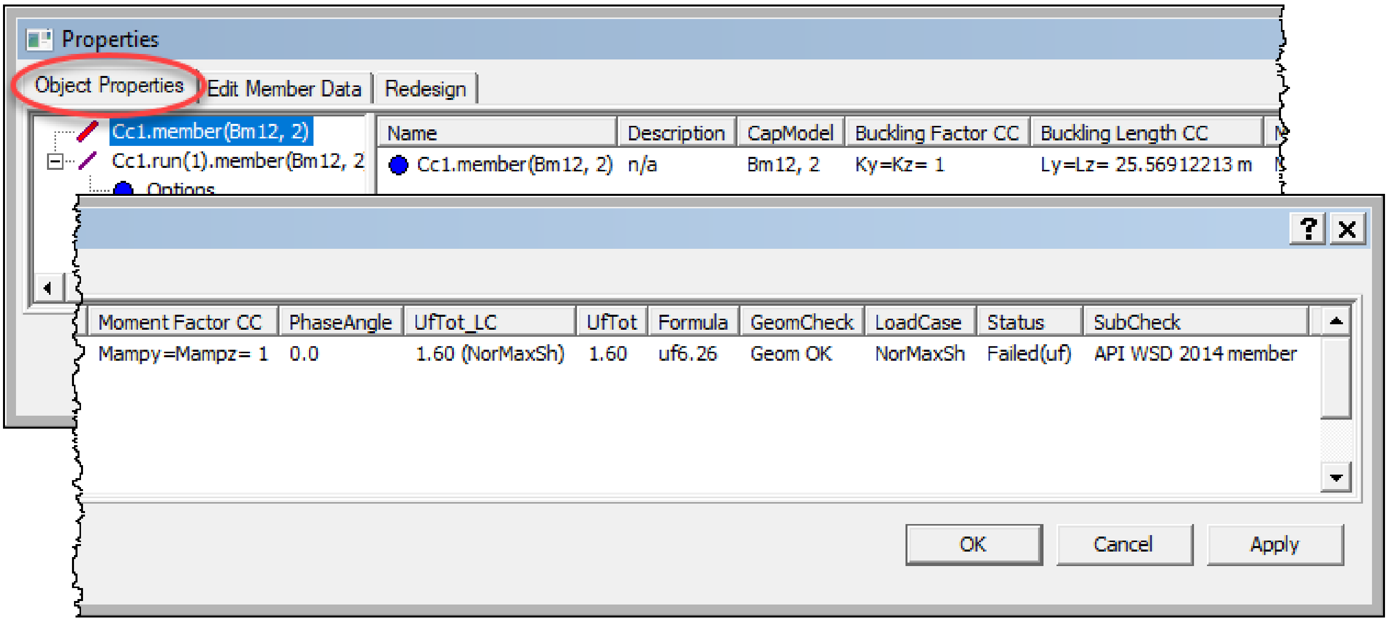


8 DETAILS FOR, AND REDESIGNING CAPACITY MEMBER

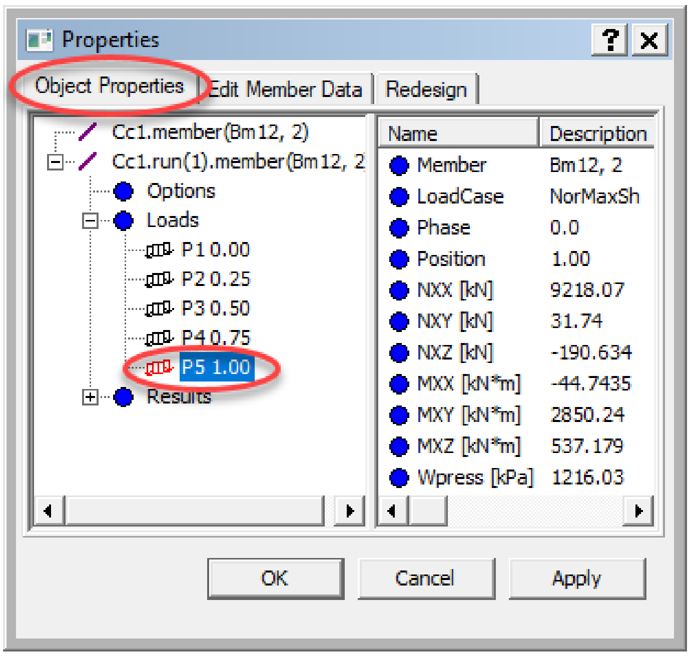
➢ See more details for a selected member by right-clicking and clicking *Properties*.



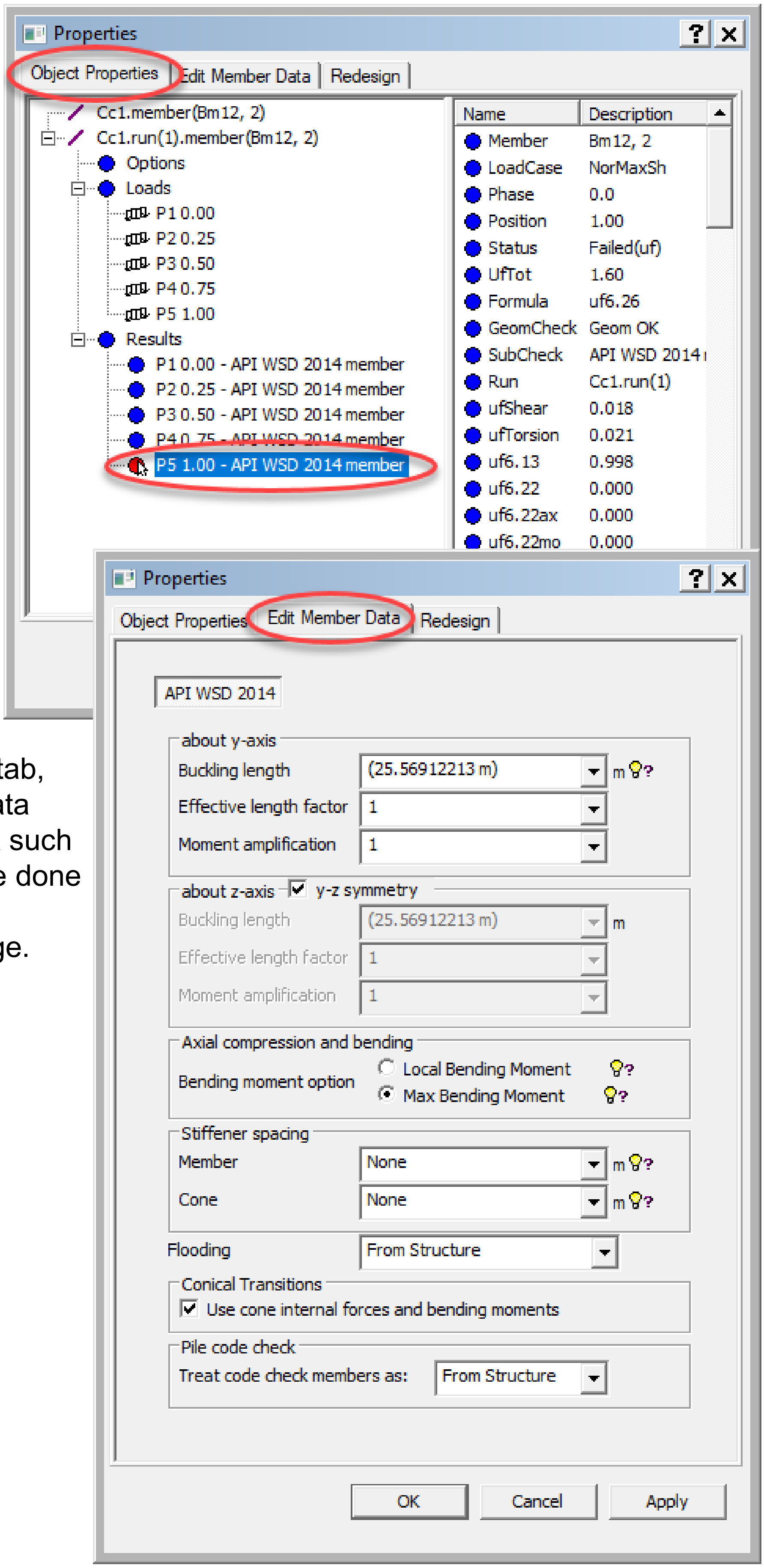
- In the *Object Properties* tab, find buckling data and code check results for the member.



- In the *Object Properties* tab, expand *Loads* and select a position to see the forces and moments at that position.

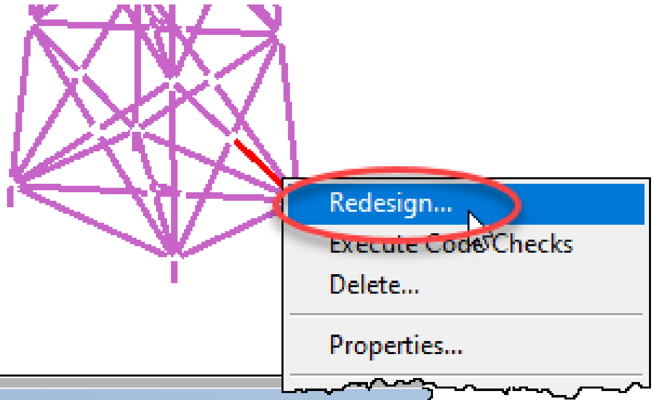


- In the *Object Properties* tab, expand *Results* and select a position to see all code check results at that position. I.e. not only the highest usage factor *UfTot* but usage factor for any equation of relevance in the code.

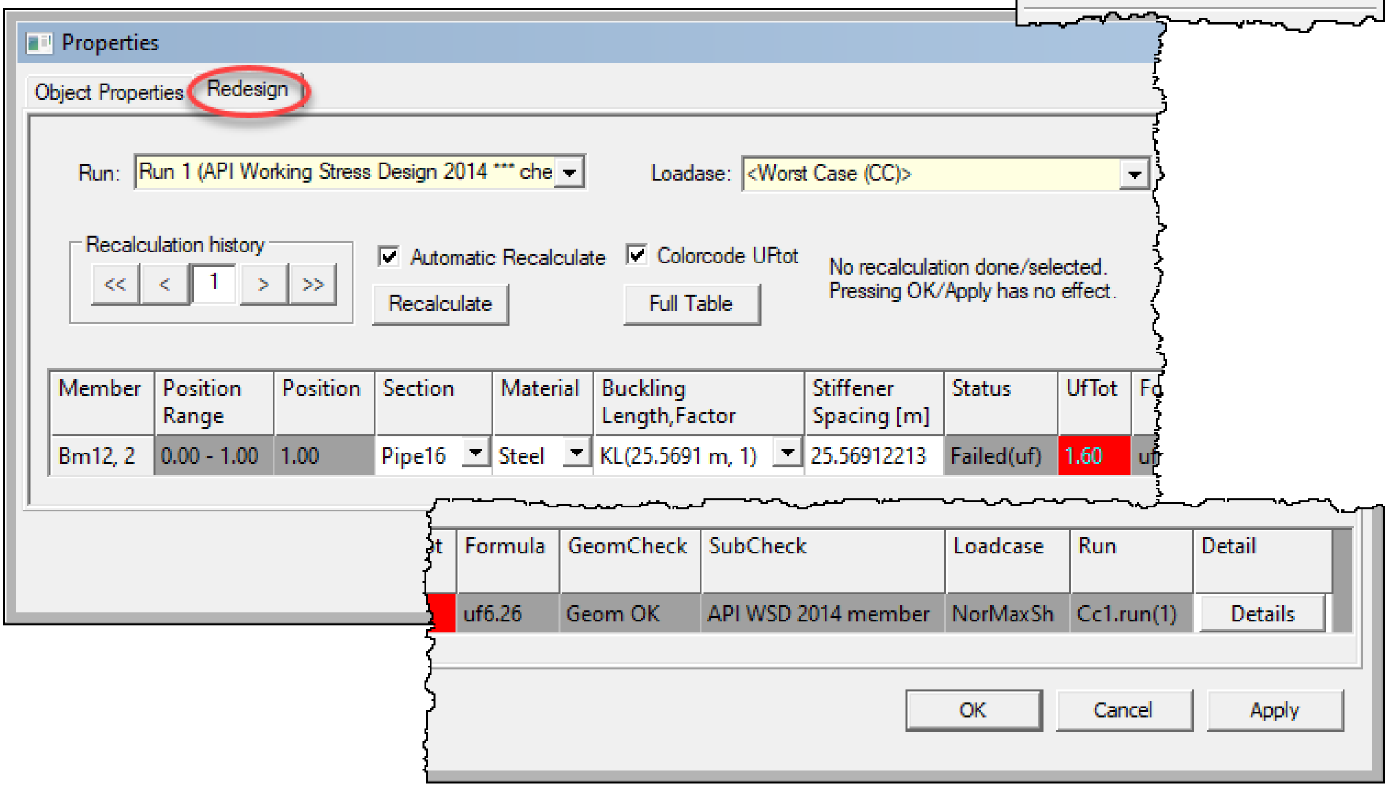


- In the *Edit Member Data* tab, find buckling data. The data may be modified here but such modifications may also be done in the *Redesign* tab as explained on the next page.

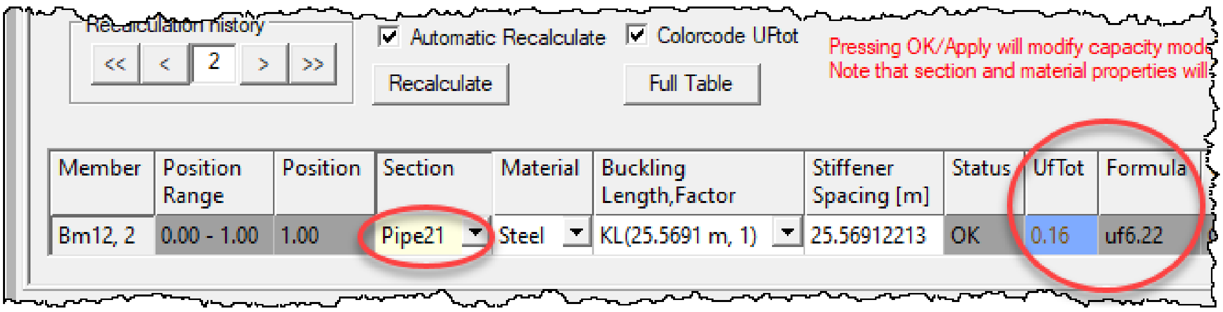
➢ Redesign of a member to ensure it isn't failing is done in the *Redesign* tab of the *Properties* dialog. The same tab may be opened by right-clicking the member and selecting *Redesign*.



- As seen, the *UfTot* is 1.60 and the critical equation is *uf6.26* which is axial tension and hydrostatic pressure.



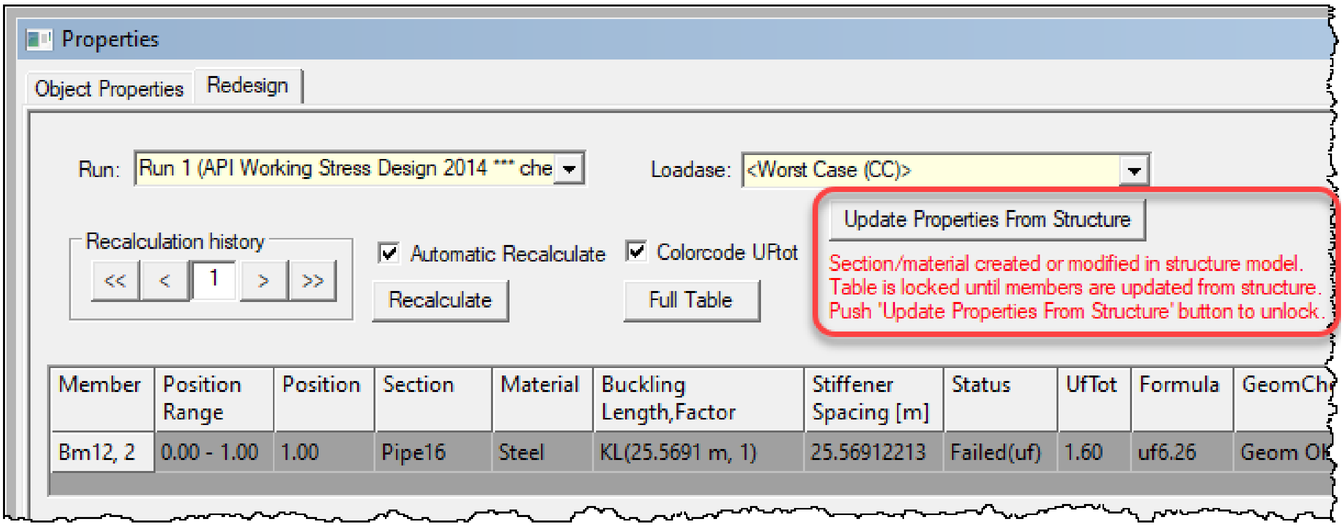
- It might be an idea to use a stronger section like Pipe21. Make this change in the dialog and see that the member no longer fails. In fact *uf6.22* – equation (6.22), i.e. axial compression and bending – is now the largest usage factor.



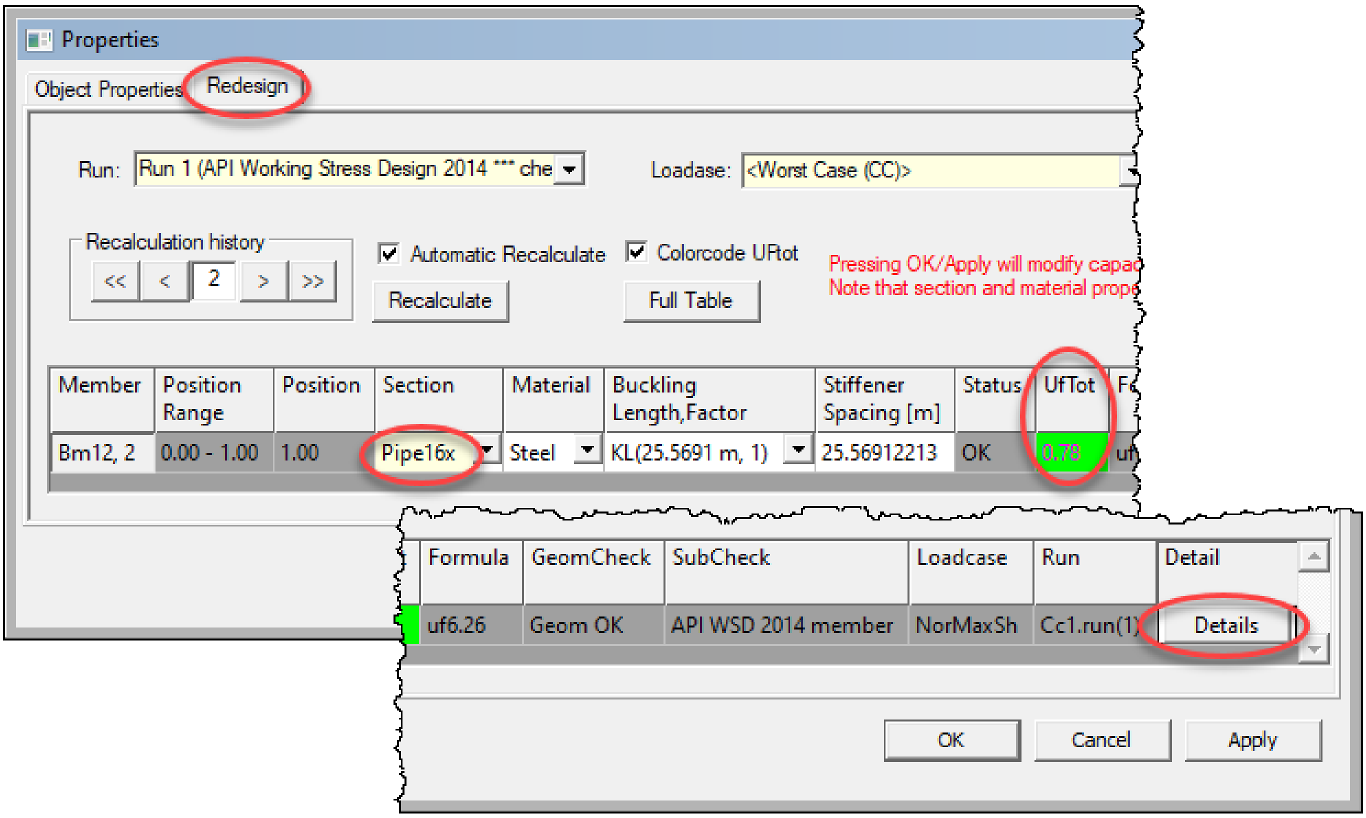
- If we rather want to use a pipe section with the same diameter as the original (1.6 m) and only increase the thickness from 3 cm to 3.5 cm then this pipe section must be created. Name it for instance Pipe16x.

➢ Having created the new pipe section return to redesigning the failing member.

  ○ Notice the text in red. This means that *Update Properties From Structure* must be clicked to make the new pipe section available to the redesign.



  ○ Having clicked the button select the new pipe section and see that the member no longer fails.



  ○ Click *Details* to see all to see all possible details. The data are tab separated to allow copy-pasting into e.g. Excel.

- Data from *Details* pasted into Excel.

Code check details for Bm12, 2	
ufShear	0.0157734
ufTorsion	0.0181579
uf6.13	0.628634
uf6.22	0
uf6.22ax	0
uf6.22mo	0
uf6.21	0.426147
uf6.21ax	0.250788
uf6.21mo	0.175359
uf6.23	0
uf6.23ax	0
uf6.23mo	0
uf6.27	0
uf6.27ax	0
uf6.27mo	0
uf6.28	0
uf6.29	0
uf6.26	0.777427
uf6.24	0
uf6.25	0
D/t	45.7143
thk	0.035
relpos	0.99999
D	1.6
t	0.035
Fy	356000
E	2.10E+08
P	9218.07
My	2850.24
Mz	537.179
Mt	44.7435

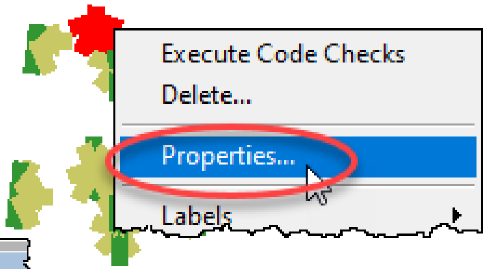
V	193.258
p	1216.03
Kly	25.5691
Klz	25.5691
stfspace	25.5691
Fey'	506637
Fez'	506637
fa	53568.3
fby	43259.7
fbz	8153.06
fv	2246.13
fvt	339.548
fbymax	43259.7
fbzmax	8153.06
Cmy	0
Cmz	0
Ft	213600
Fxe	2.76E+06
Fxc	356000
Fa	177930
Fb	251036
Fv	142400
Fvt	142400
fh	27795
Fhe	88429.7
Fhc	88429.7
SFh	2
SFxt	1.67
SFxc	1.81741
SFb	1.41813
sFac	1

- Click *OK* or *Apply* in the *Redesign* tab to commit the change of section to Pipe16x.

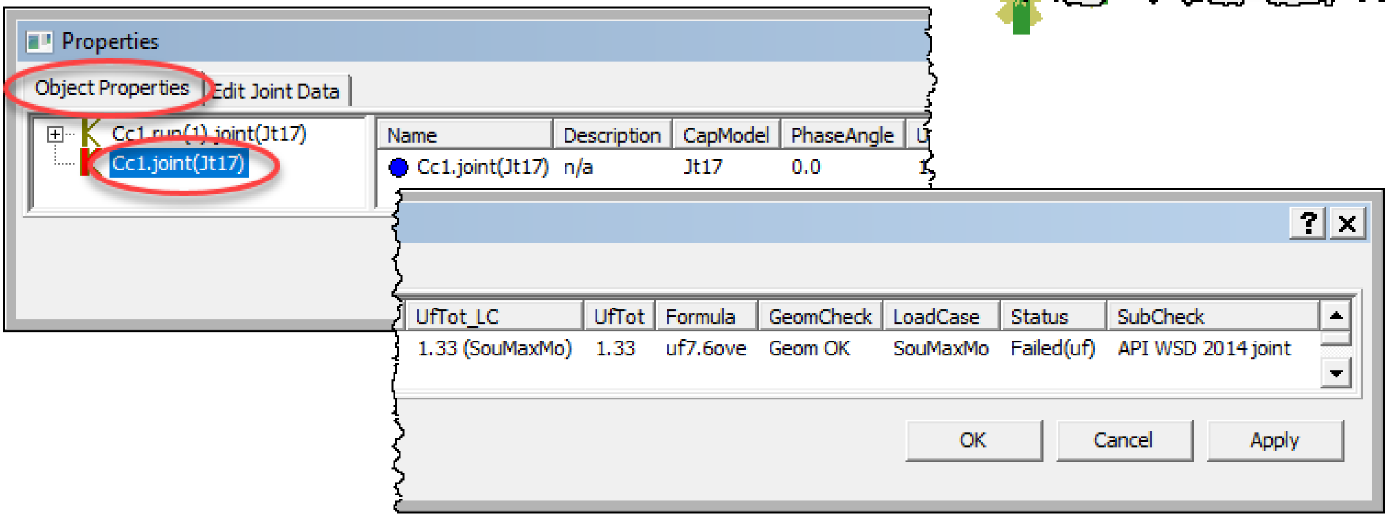
➢ Note, however, that this redesign has been done based on the assumption that the forces and moments in the redesigned member will not change when shifting to another pipe section. This is of course not the case. A stronger beam will redistribute forces in the whole model causing the forces and moments of the redesigned member to increase somewhat.

9 DETAILS FOR, AND REDESIGNING CAPACITY JOINT

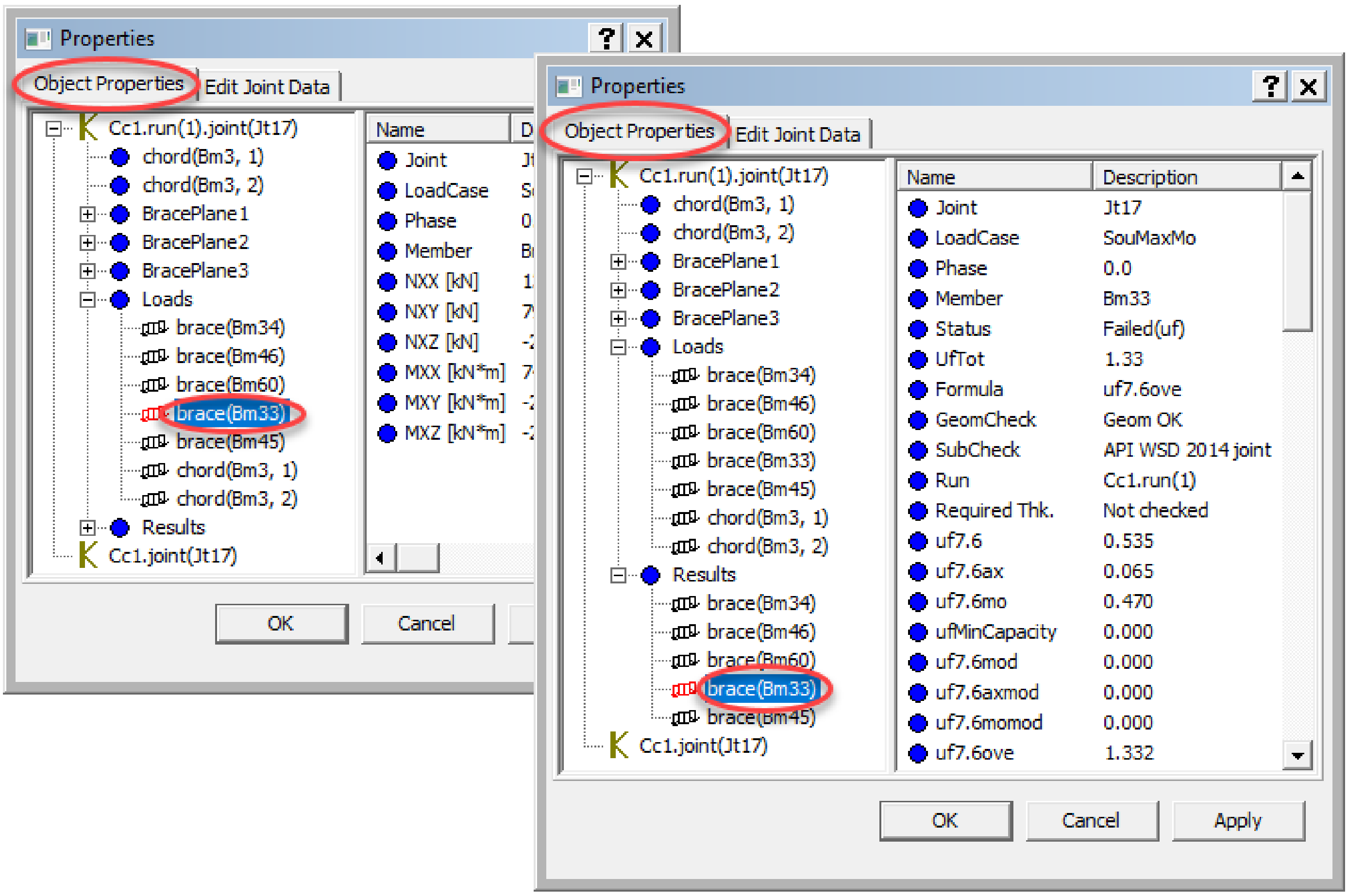
- See more details for a selected joint by right-clicking and clicking *Properties*.



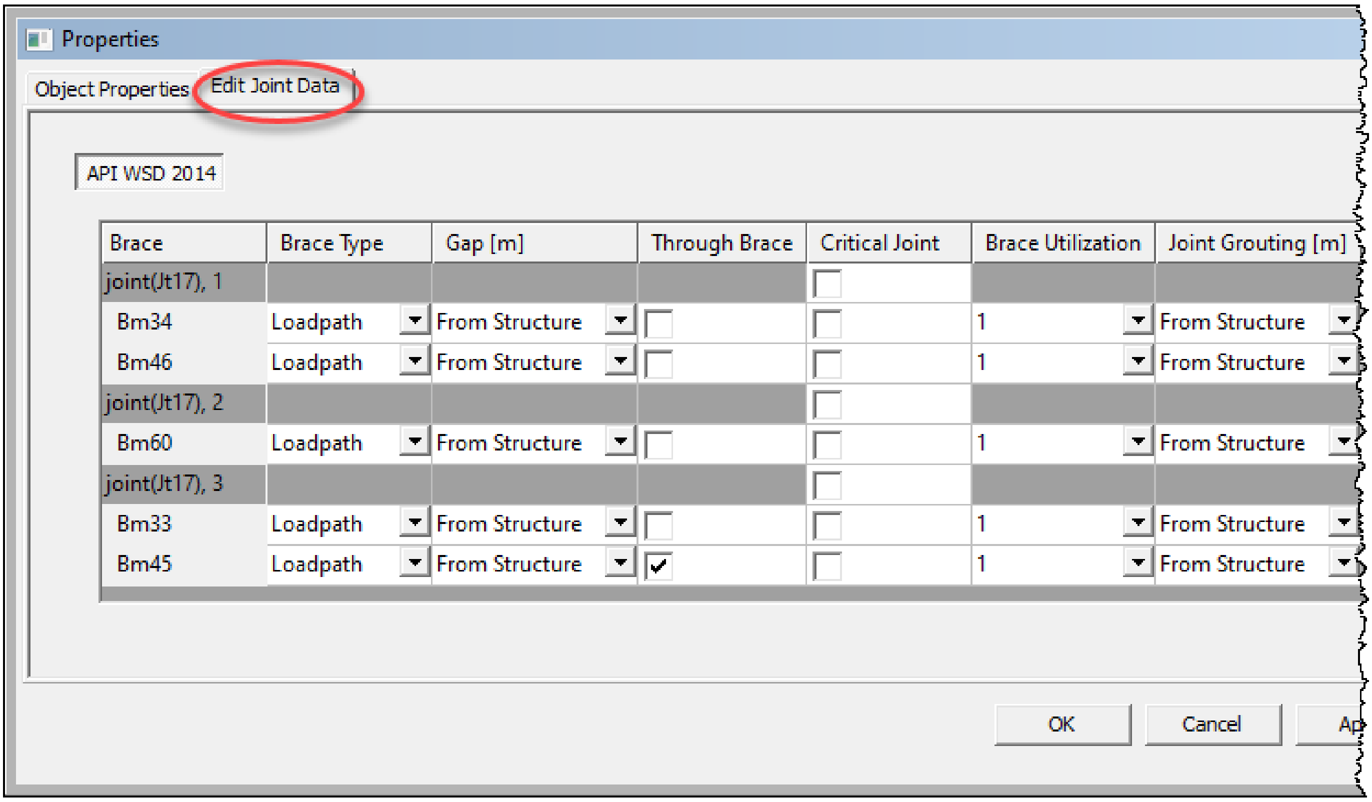
  - In the *Object Properties* tab, find code check results for the joint.



  - In the *Object Properties* tab, expand *Loads* and select a brace to see the forces and moments in that brace. Expand *Results* and select a brace to see all code check results for that brace. I.e. usage factor for any equation of relevance.

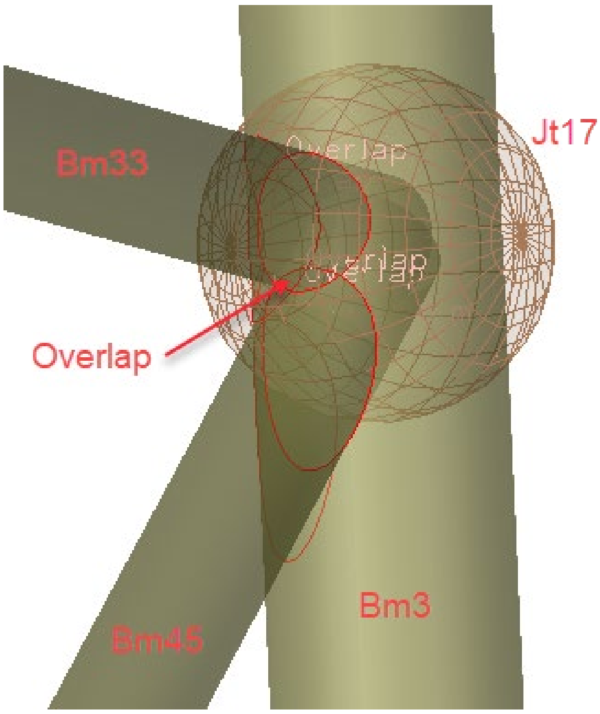


- In the *Edit Joint Data* tab of the *Properties* dialog see and modify joint data on which the code checking is based.

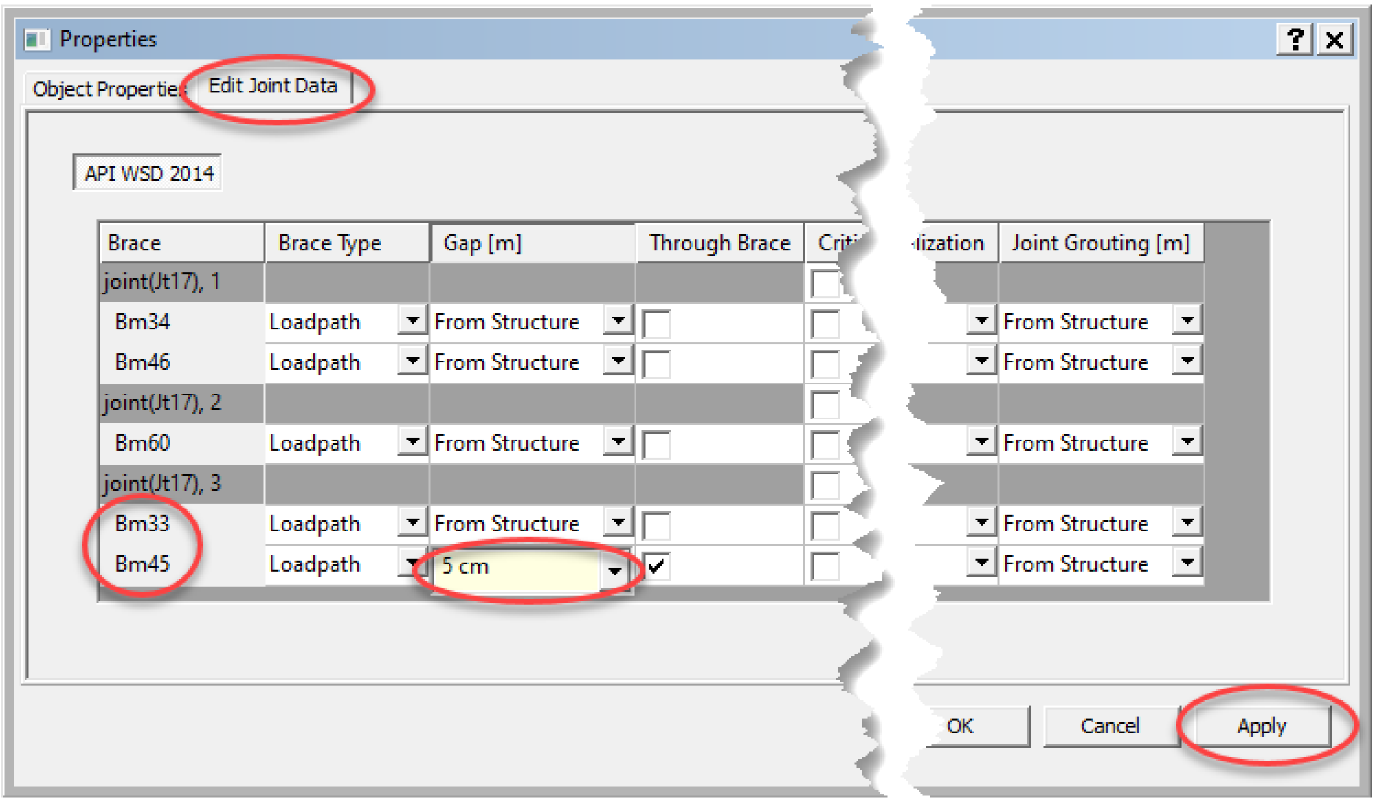


Brace	Brace Type	Gap [m]	Through Brace	Critical Joint	Brace Utilization	Joint Grouting [m]
joint(Jt17), 1				☐		
Bm34	Loadpath	From Structure	☐	☐	1	From Structure
Bm46	Loadpath	From Structure	☐	☐	1	From Structure
joint(Jt17), 2				☐		
Bm60	Loadpath	From Structure	☐	☐	1	From Structure
joint(Jt17), 3				☐		
Bm33	Loadpath	From Structure	☐	☐	1	From Structure
Bm45	Loadpath	From Structure	☑	☐	1	From Structure

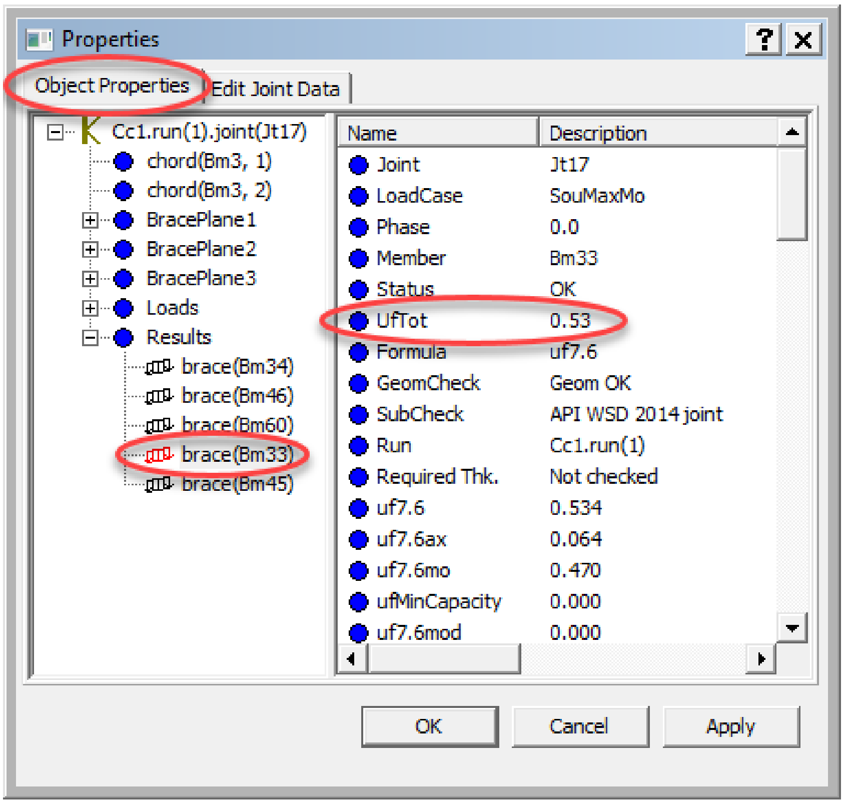
- As shown on the previous page, brace Bm33 fails for formula *uf7.6ove* which means 'Usage factor from overlap brace in overlapping joint, through brace as chord'.
- From the *Edit Joint Data* tab above is seen that Bm45 is the through brace, i.e. Bm33 is cut to fit both the leg Bm3 and brace Bm45 and welded onto these.
- Double-click a joint (the brown ball) to see the intersection lines between the chord and braces. In the figure to the right joint Jt17 has been double-clicked to reveal the overlap between the two braces. (Other braces have been removed from the display for clarity.)
- However, proper design of a tubular joint dictates that there should be a gap of 5 cm between the two braces when welded onto the leg.



- Without modifying the concept model a gap of 5 cm may be introduced in the *Edit Joint Data* tab of the *Properties* dialog as shown below. First click the pull-down menu to change *From Structure* into *Manual* and then type in 5 cm.
- Click *Apply*.

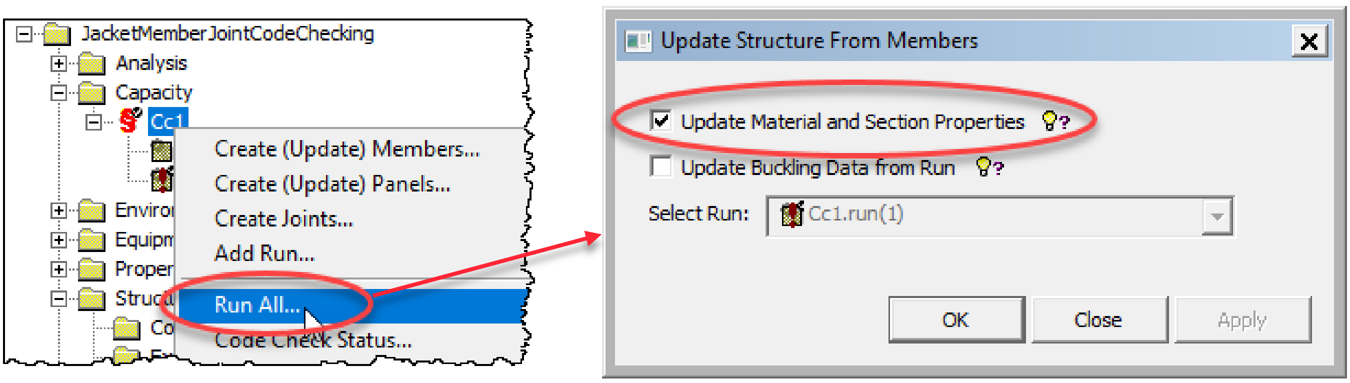


- Go to the *Object Properties* tab and see that *UfTot* has been reduced and the joint no longer fails as seen to the right.
- However, this redesign has been done based on the assumption that the forces and moments in the joint will not change when introducing a gap. This is of course not the case. A gap will redistribute forces in the whole model causing the forces and moments in the joint to change somewhat.

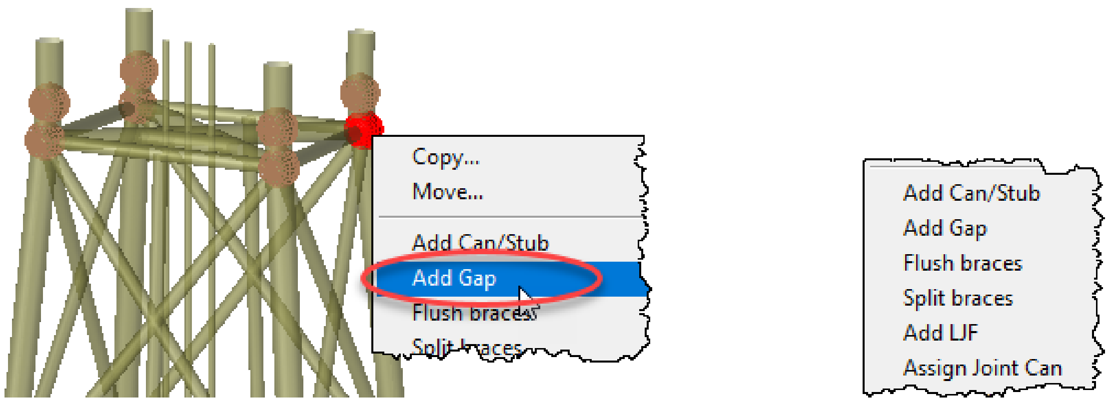


10 RE-RUN THE STRUCTURAL ANALYSIS AND CODE CHECKING

- As pointed out above, changing pipe sections for members and introducing gaps in tubular joints will redistribute forces in the whole model causing the forces and moments in members and joints to change.
- Therefore re-running the structural analysis followed by a new code checking is required to confirm the adequacy of the redesigns.
- Right-click the capacity manager in the browser and select *Run All* as shown below to:
  - Transfer all member redesigns, typically changes of sections, from the capacity model to the concept model. This is done when *Update Material and Section Properties* in the *Update Structure From Member* dialog is checked.
  - Re-run the structural analysis.
  - Re-run the code checking.



- However, gaps introduced in capacity joints will not be transferred to the concept model. Gaps must therefore be introduced in the concept model as shown below. This involves introducing eccentricities to braces to ensure proper gaps. Possibly also cans and stubs, flushing braces to the chord wall and other joint design measures must be taken. And not only to the joint shown but to other joints as well. The context sensitive menu to the right below shows joint design options.



- After proper design of all joints, right-click the capacity manager in the browser and select *Run All* as shown above.

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