

SESAM TUTORIAL

GeniE

Finding Centre of Force (CoF) and Redesigning by Editing Beam Cross Section Properties

Valid from program version 8.2



Sesam Tutorial

GeniE – Finding Centre of Force (CoF) and Redesigning by Editing Section Properties

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 AS. All rights reserved

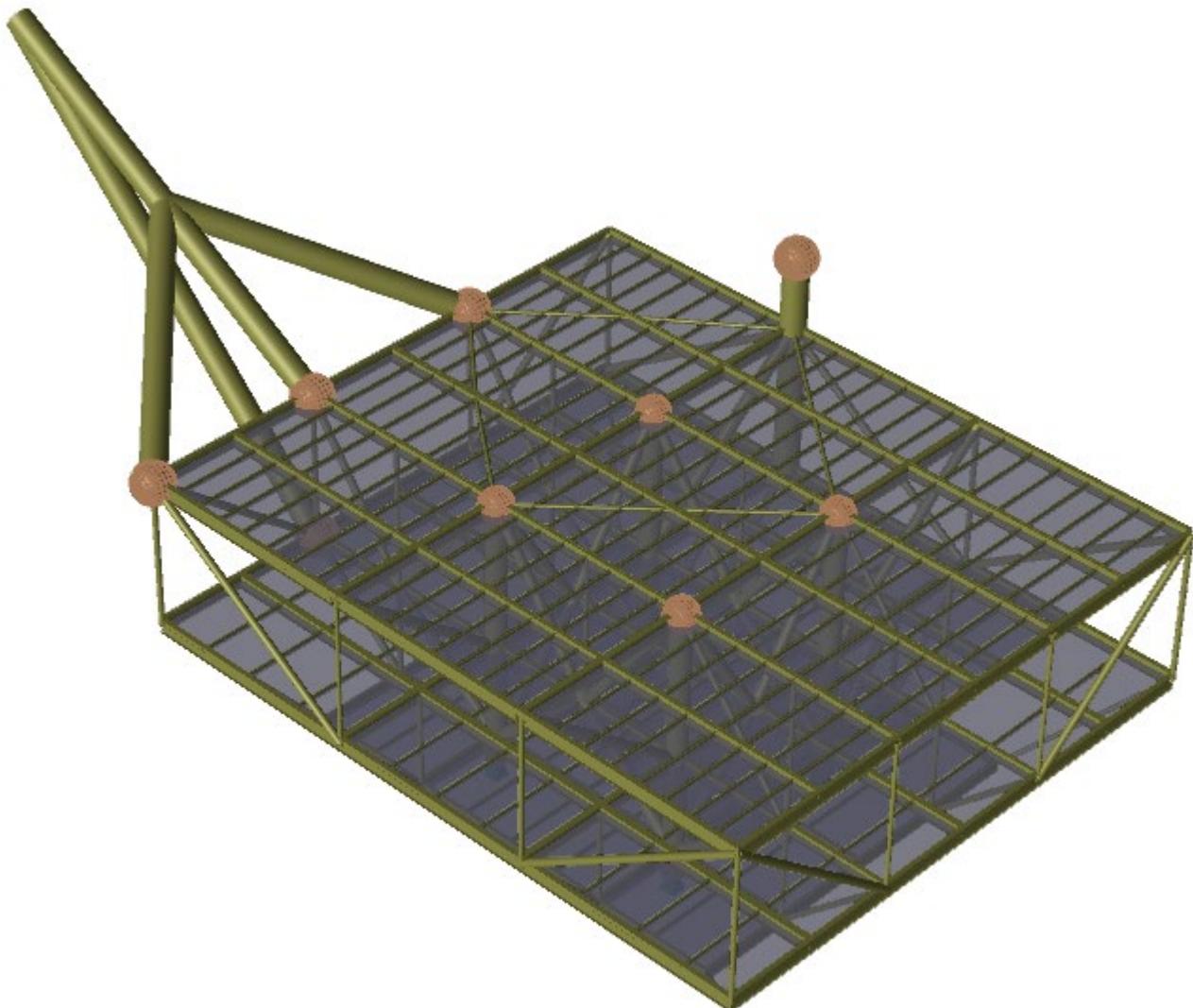
This publication or parts thereof may not be reproduced or transmitted in any form or by any means, including copying or recording, without the prior written consent of DNV AS.

TABLE OF CONTENTS

1. Introduction	Page 4
2. Open New Workspace	Page 5
3. Import Existing Model	Page 6
4. Run Analysis	Page 7
5. Find Centre of Force	Page 8
6. Initial Code Checking	Page 11
7. Redesign by Editing Beam Cross Section Properties	Page 12

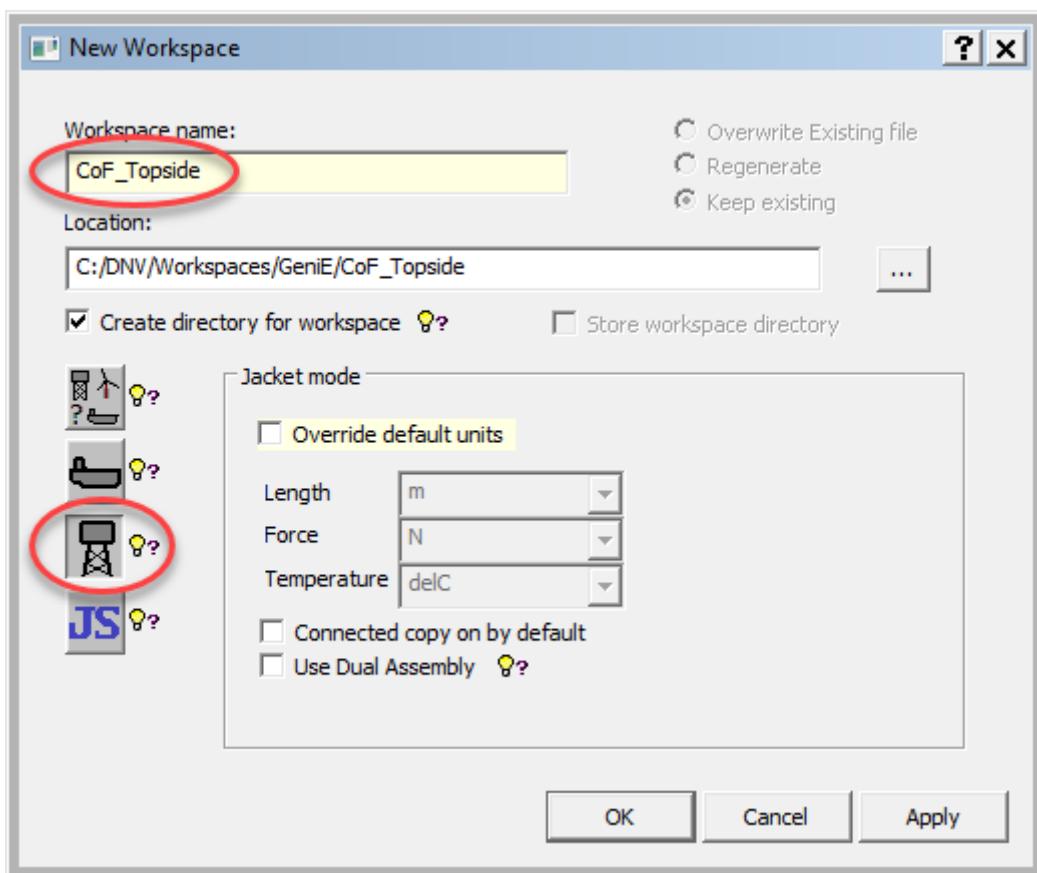
1 INTRODUCTION

- This tutorial demonstrates how to find the centre of force (CoF) for individual load cases and load combinations. The CoF calculation includes structural weight, equipment forces, point loads, line loads and surface loads (pressure).
- Moreover, the tutorial shows how to edit section properties and the effect of such in structural analysis and code check.
- There is no modelling in this tutorial as an existing model is imported. Go to other tutorials for learning about modelling in GeniE.



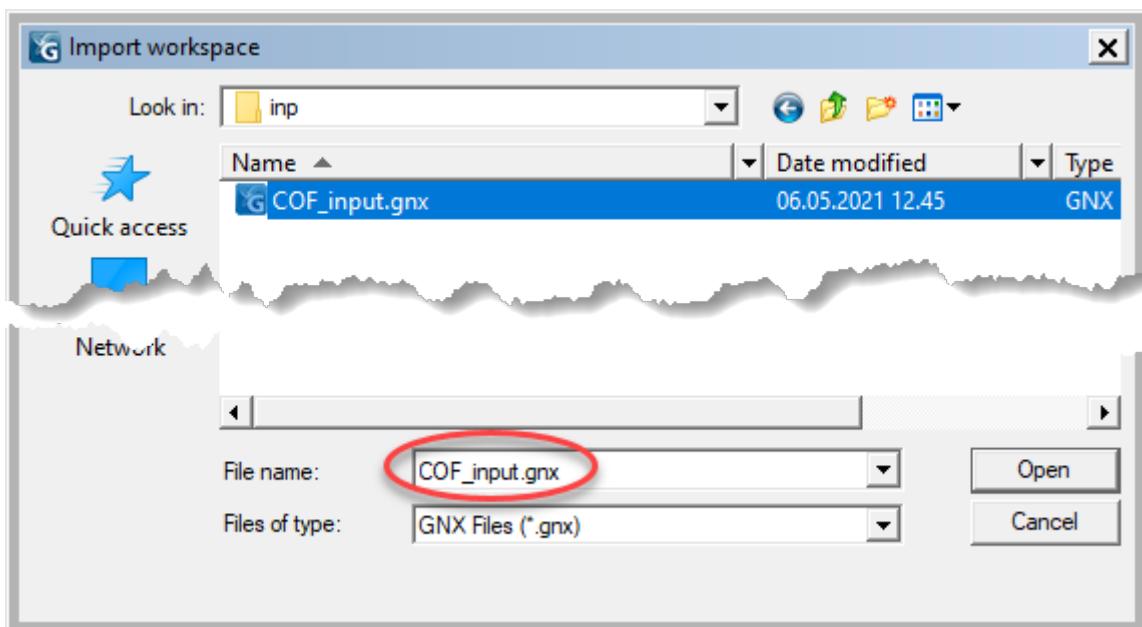
2 OPEN NEW WORKSPACE

- 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 (spaceframe) modelling.
 - If the workspace name exists, select *Overwrite Existing file* or give another name.

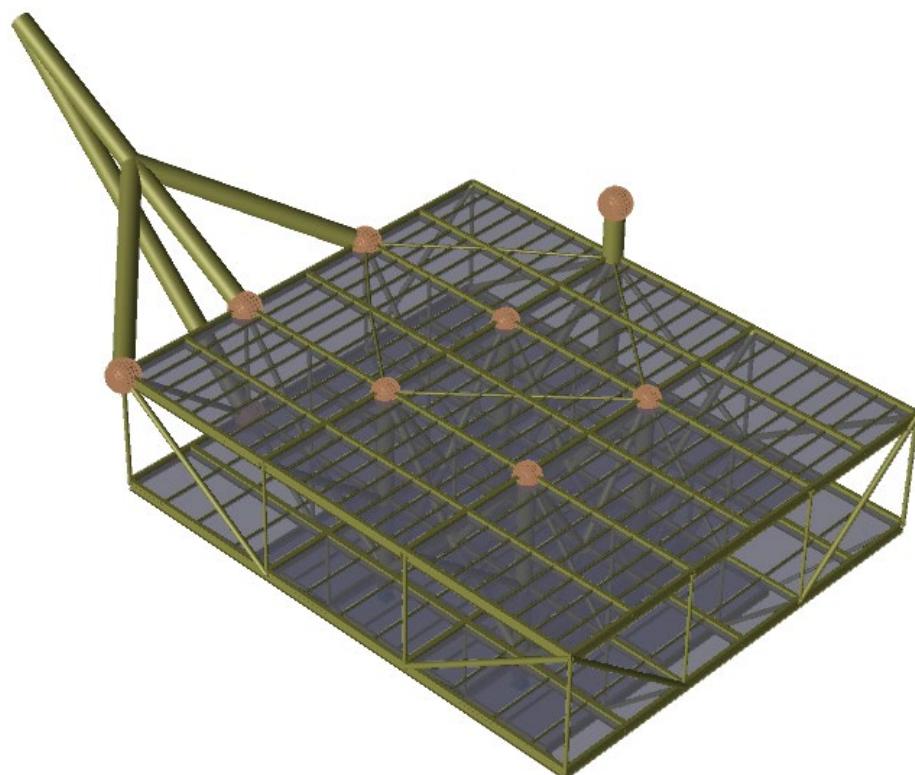


3 IMPORT EXISTING MODEL

- Use *File | Import | Workspace (GNX file)* to import an existing model.
- Find the GNX file COF_input.gnx in the installation folder:
<path>\GeniE VX.Y-ZZ\Help\Tutorials\TutorialsBasicAndCodechecking\B12_Finding_Center_of_Force\inp
- Click *Open* and then F5 (Iso view) to fit the model in the display area.

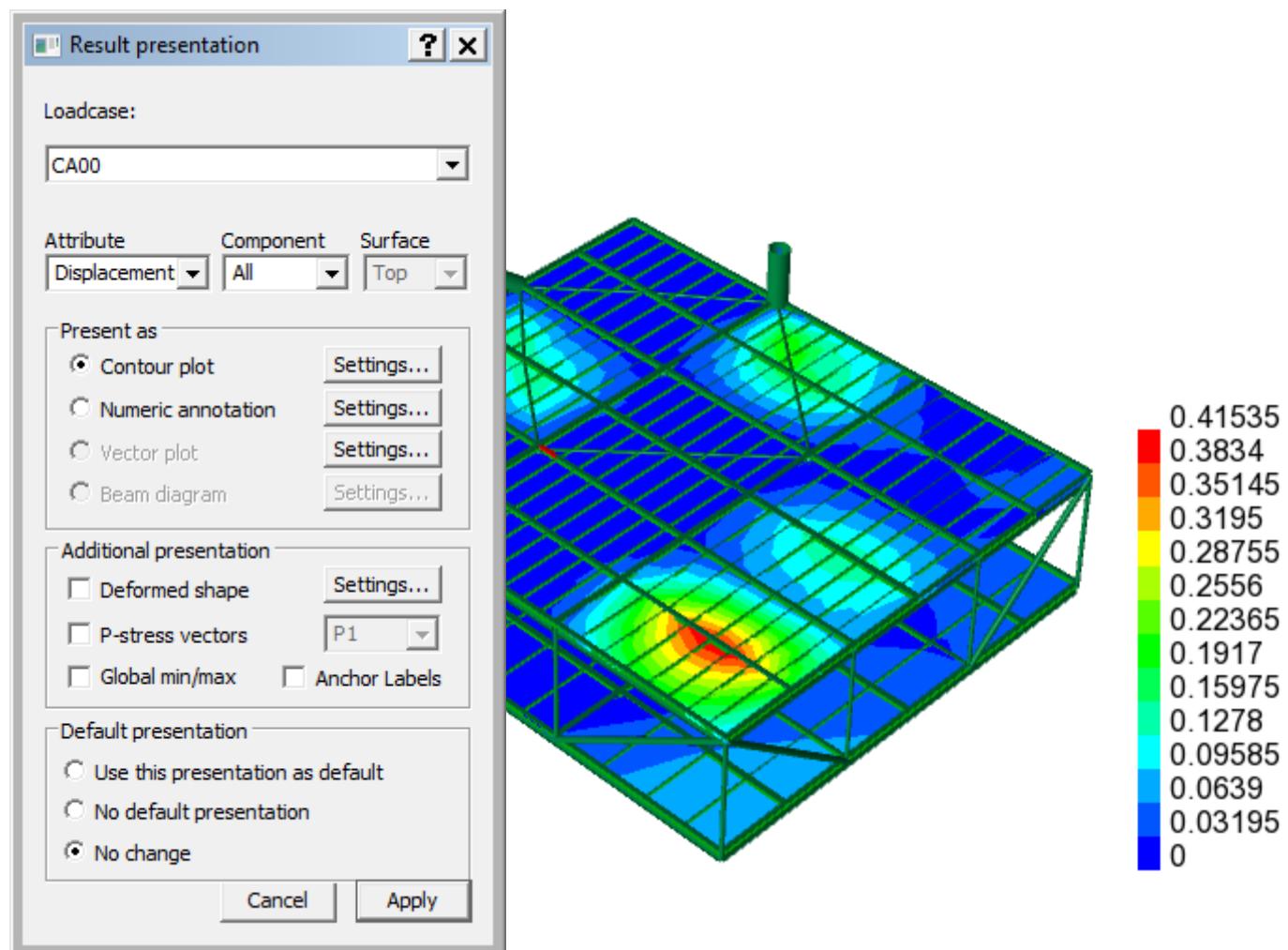


- The model:



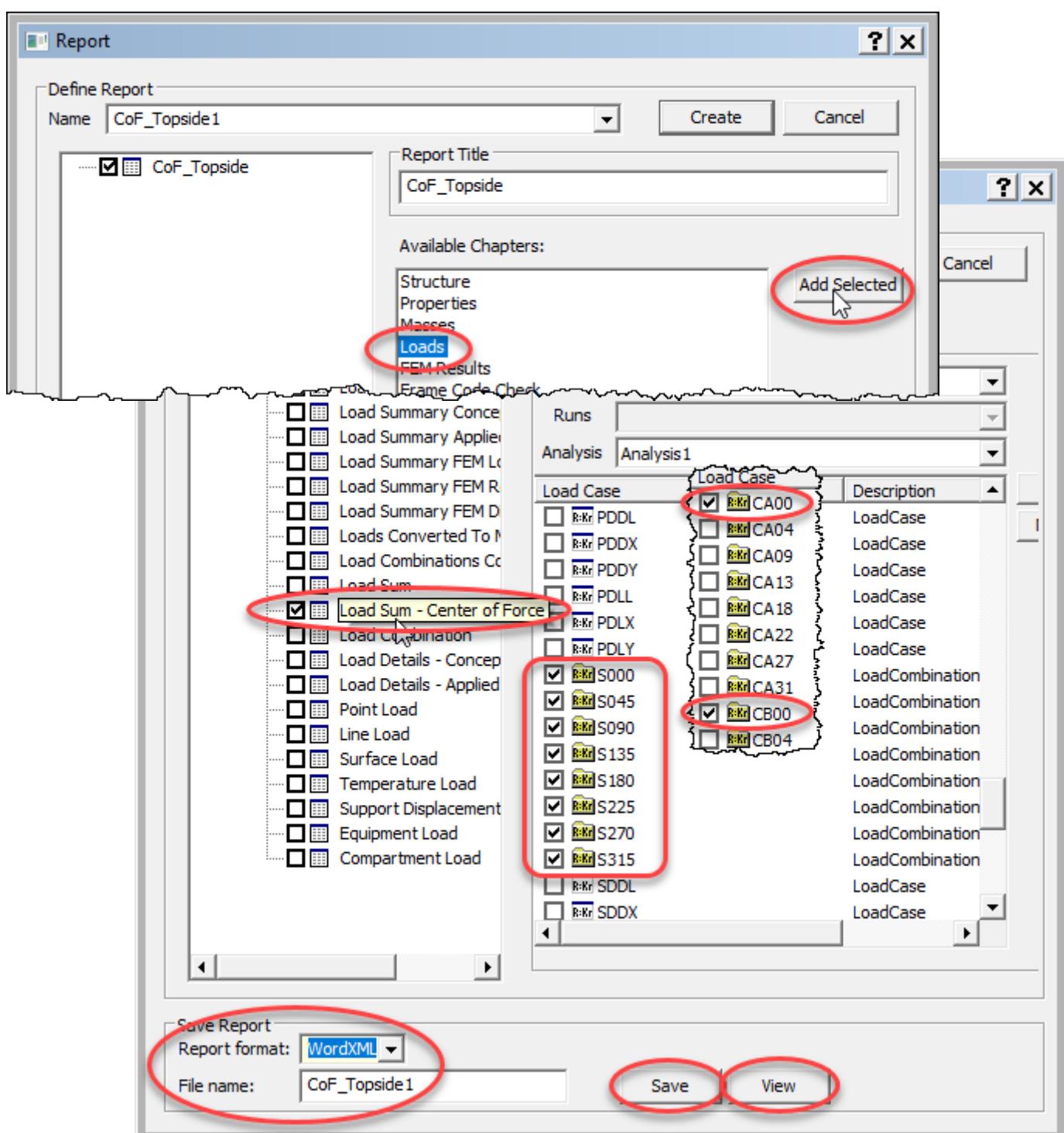
4 RUN ANALYSIS

- The imported model contains load cases, load combinations and a static linear structural analysis activity.
- Run the analysis by *Mesh & Analysis | Activity Monitor* (or Alt+D) and clicking *Start*.
- View some result by switching to *Results - with Mesh* display configuration and using *Results | Presentation* to select a load case, attribute and component as shown below.



5 FIND CENTRE OF FORCE

- The centre of force is found in a report produced by *File | Save report*.
- In the *Report* dialog add the chapter *Loads* to the report.
- Thereafter select only the sub-chapter *Load Sum – Center of Force* and the load cases (combinations) CA00, CB00, S000, S045, S090, S135, S180, S225, S270 and S315.
- Select format, e.g. *WordXML*. Click *Save* to create the report and *View* to open it.



- Having opened the report in Word, right-click in the table of contents and select *Update Field* to update the page numbers.

The screenshot shows a Microsoft Word document with a header containing the GeniE logo and version information (GeniE V8.1-37), and model details (Model Id: CoF_Topside1, Description: CoF_Topside, Model file name: C:/DNV/GL/Workspaces/GeniE/CoF_Topside). Below the header is the 'Table of Contents' section. A context menu is open over the table of contents, specifically over the entry '1.2.1 CA00 : Load Sum - Center of Force'. The menu items include Cut, Copy, Paste Options, Update Field (which is highlighted with a red circle), Edit Field..., and Toggle Field Codes. Above the context menu, the Word ribbon is visible, showing the font 'Times New Roman' and size '8'. A large white arrow points downwards from the context menu towards the bottom of the screen.

Section	Page Number
1 Loads	3
1.1 All loadcases	3
1.2 CA00	4
1.2.1 CA00 : Load Sum - Center of Force	4
1.3 CB00	5
1.3.1 CB00 : Load Sum - Center of Force	5
1.4 S000	6
1.4.1 S000 : Load Sum - Center of Force	6
1.5 S045	7
1.5.1 S045 : Load Sum - Center of Force	7
1.6 S090	8
1.6.1 S090 : Load Sum - Center of Force	8
1.7 S135	9
1.7.1 S135 : Load Sum - Center of Force	9
1.8 S180	10
1.8.1 S180 : Load Sum - Center of Force	10
1.9 S225	11
1.9.1 S225 : Load Sum - Center of Force	11
1.10 S270	12
1.10.1 S270 : Load Sum - Center of Force	12
1.11 S315	13
1.11.1 S315 : Load Sum - Center of Force	13

➤ For the load combination CA00 find the following table:

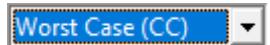
LoadCase	FEM LC	LoadType	FX [kN]	FY [kN]	FZ [kN]	MX [kN*m]	MY [kN*m]	MZ [kN*m]	Direction	CofX [m]	CofY [m]	CofZ [m]	Notes
CA00	39	Explicit Point	0	0	-2330.87	-24239.2	-26475.5	0	FZ	-16.8361	10.3992	37.3598	
									FX				Zero
									FY				Zero
CA00	39	Explicit Line	0	0	-43268.9	-3.72529e-12	-48.643	0	FZ	-0.0011242	0	23.9177	
									FX				Zero
									FY				Zero
CA00	39	Explicit Surface	0	0	-41722.8	45362.7	201166	0	FZ	4.82148	-1.08724	30.48	
									FX				Zero
									FY				Zero
CA00	39	Explicit Conceptual	0	0	-87322.6	21123.5	174641	0	FZ	1.85375	-0.241902	27.412	
									FX				Zero
									FY				Zero
CA00	39	Explicit Applied	0	0	-87322.6	21123.5	174641	0	FZ	1.85375	-0.241902	27.412	
									FX				Zero
									FY				Zero
CA00	39	Equipment Conceptual											
CA00	39	Equipment Applied											
CA00	39	Structure Conceptual	0	0	-8609.5	-5611.05	-2355.45	0	FZ	-0.273587	0.651728	23.5892	
									FX				Zero
									FY				Zero
CA00	39	Total Conceptual	0	0	-95932.1	15512.4	172286	0	FZ	1.66283	-0.161702	27.0689	
									FX				Zero
									FY				Zero
CA00	39	Total Applied	0	0	-95932.1	15512.4	172286	0	FZ	1.66283	-0.161702	27.0689	
									FX				Zero
									FY				Zero

➤ See that the CoFs for individual loads contributing to CA00 are given and the CoF for the total load is (1.66823 m, -0.161702 m, 27.0689 m).

➤ In Excel format the same table appears like this:

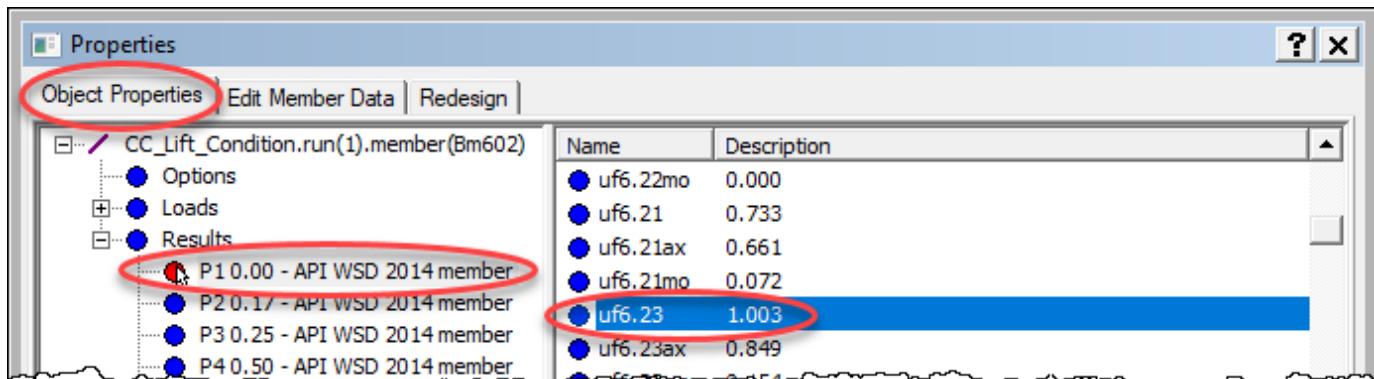
A	B	C	D	E	F	G	H	I	J	K	L	M	N	
1	LoadCase	FEM LC	LoadType	FX [kN]	FY [kN]	FZ [kN]	MX [kN*m]	MY [kN*m]	MZ [kN*m]	Direction	CofX [m]	CofY [m]	CofZ [m]	Notes
2	CA00	39	Explicit Point	0	0	-2330.87	-24239.2	-26475.5	0	FZ	-16.8361	10.3992	37.3598	
3										FX				Zero
4										FY				Zero
5	CA00	39	Explicit Line	0	0	-43268.9	-3.72529E-12	-48.643	0	FZ	-0.0011242	0	23.9177	
6										FX				Zero
7										FY				Zero
8	CA00	39	Explicit Surface	0	0	-41722.8	45362.7	201166	0	FZ	4.82148	-1.08724	30.48	
9										FX				Zero
10										FY				Zero
11	CA00	39	Explicit Conceptual	0	0	-87322.6	21123.5	174641	0	FZ	1.85375	-0.241902	27.412	
12										FX				Zero
13										FY				Zero
14	CA00	39	Explicit Applied	0	0	-87322.6	21123.5	174641	0	FZ	1.85375	-0.241902	27.412	
15										FX				Zero
16										FY				Zero
17	CA00	39	Equipment Conceptual											
18	CA00	39	Equipment Applied											
19	CA00	39	Structure Conceptual	0	0	-8609.5	-5611.05	-2355.45	0	FZ	-0.273587	0.651728	23.5892	
20										FX				Zero
21										FY				Zero
22	CA00	39	Total Conceptual	0	0	-95932.1	15512.4	172286	0	FZ	1.66283	-0.161702	27.0689	
23										FX				Zero
24										FY				Zero
25	CA00	39	Total Applied	0	0	-95932.1	15512.4	172286	0	FZ	1.66283	-0.161702	27.0689	
26										FX				Zero
27										FY				Zero

6 INITIAL CODE CHECKING

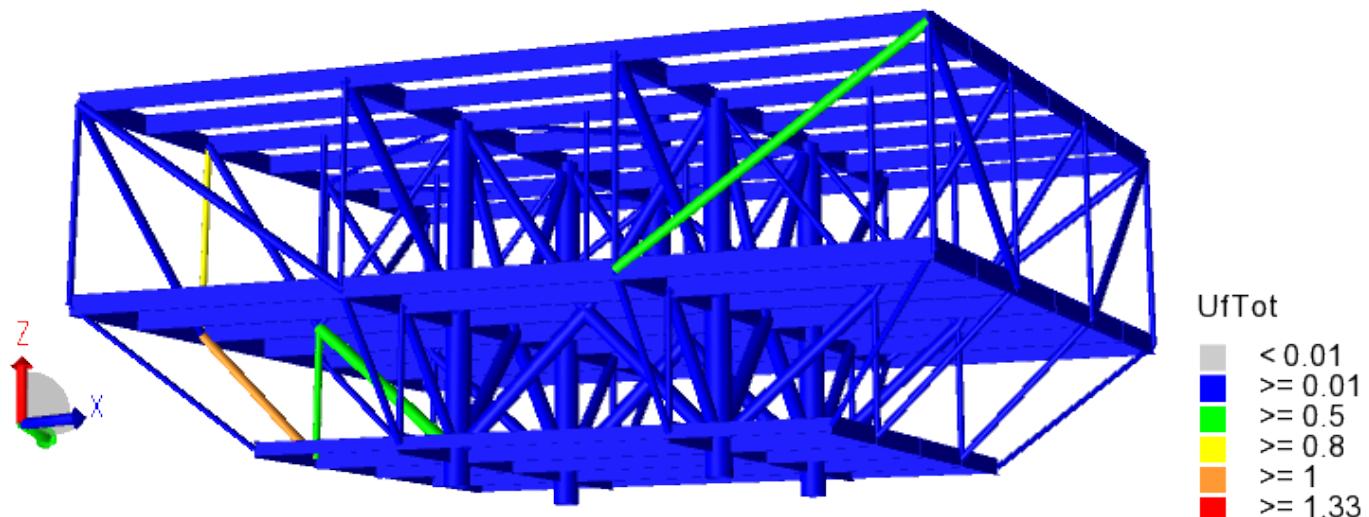
- The imported model has setup for performing code checking of three load combinations according to API WSD 2014.
 - Right-click the code check run named CC_Lift_Condition.run(1) and select *Properties* to study the code checking setup.
- Right-click the code check run and select *Execute Code Check* to perform the code checking.
- Select *Worst Case (CC)* in the load case selector: 
- See that one member (named Bm602) fails for formula *uf6.23*:

Capacity Model	LoadCase	Position	Status	UfTot	Formula	SubCheck
/ member(Bm602)	L2	0.00	Failed(uf)	1.00	uf6.23	API WSD 2014 memb
/ member(Bm568)	L2	1.00	OK	0.98	uf6.21	API WSD 2014 memb
/ member(Bm563)	L2	0.00	OK	0.80	uf6.24	API WSD 2014 memb

- Right-clicking the member in the table above and selecting *Properties* gives higher precision for the utilisation factor:

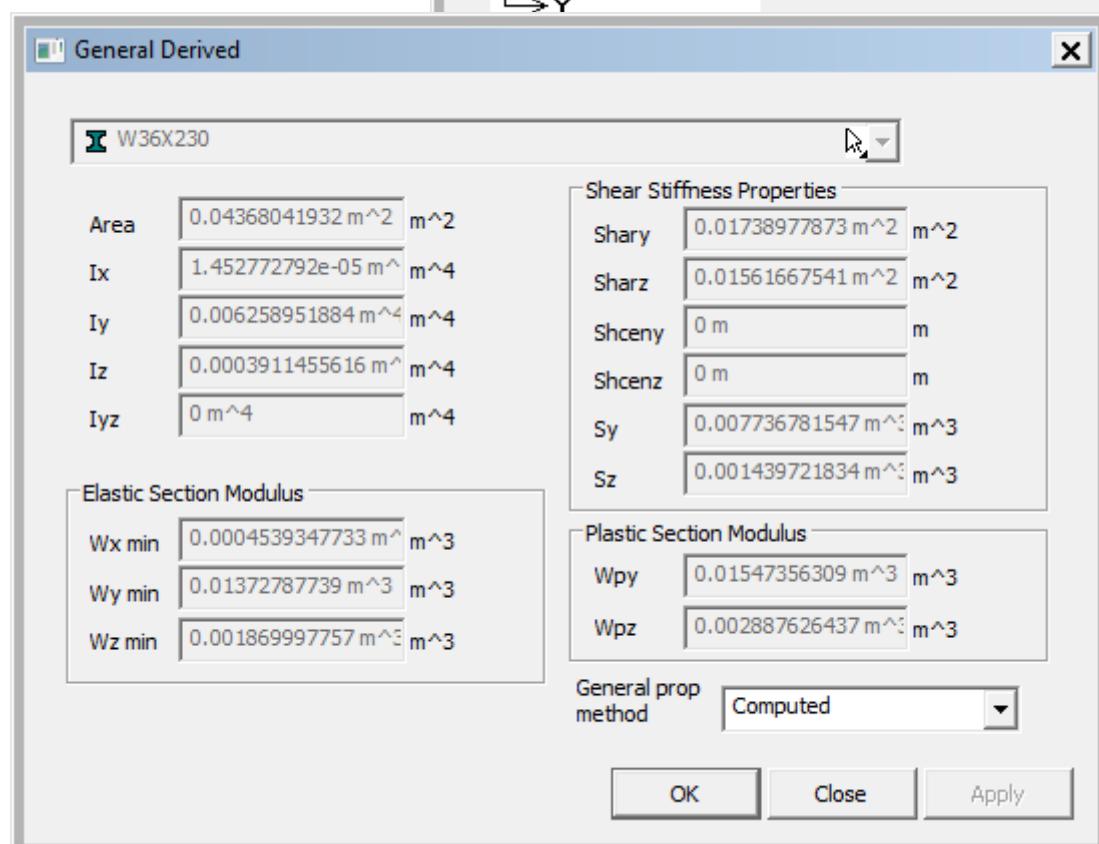
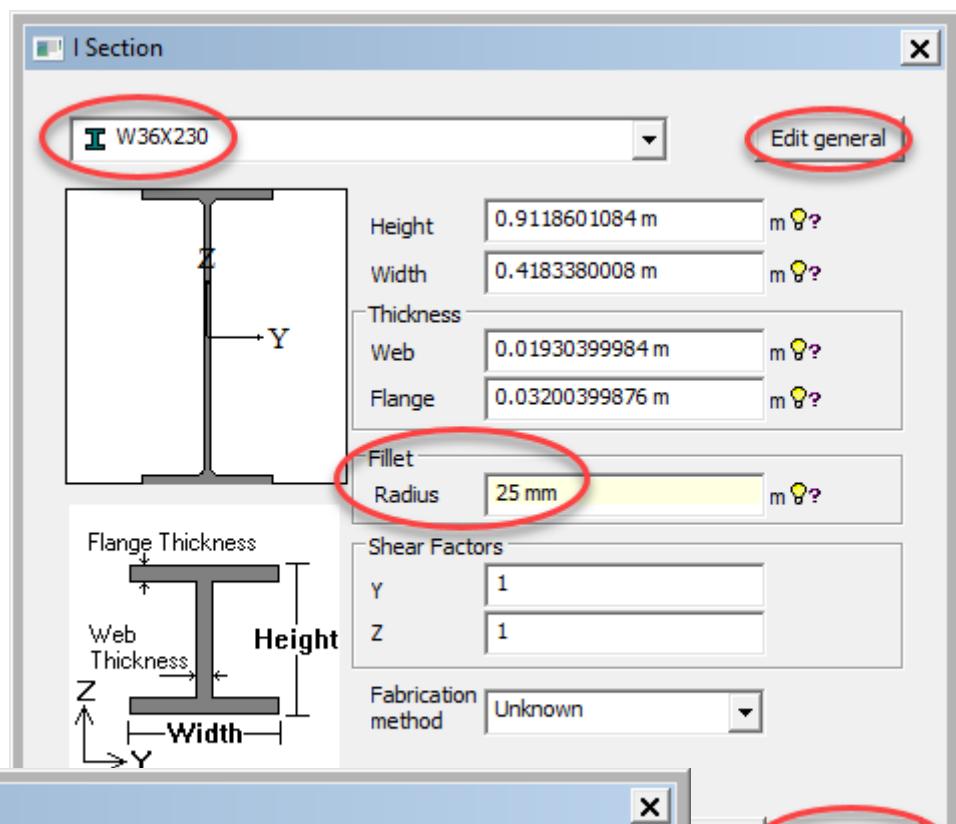


- The code check results shown graphically, failing member Bm602 is orange:

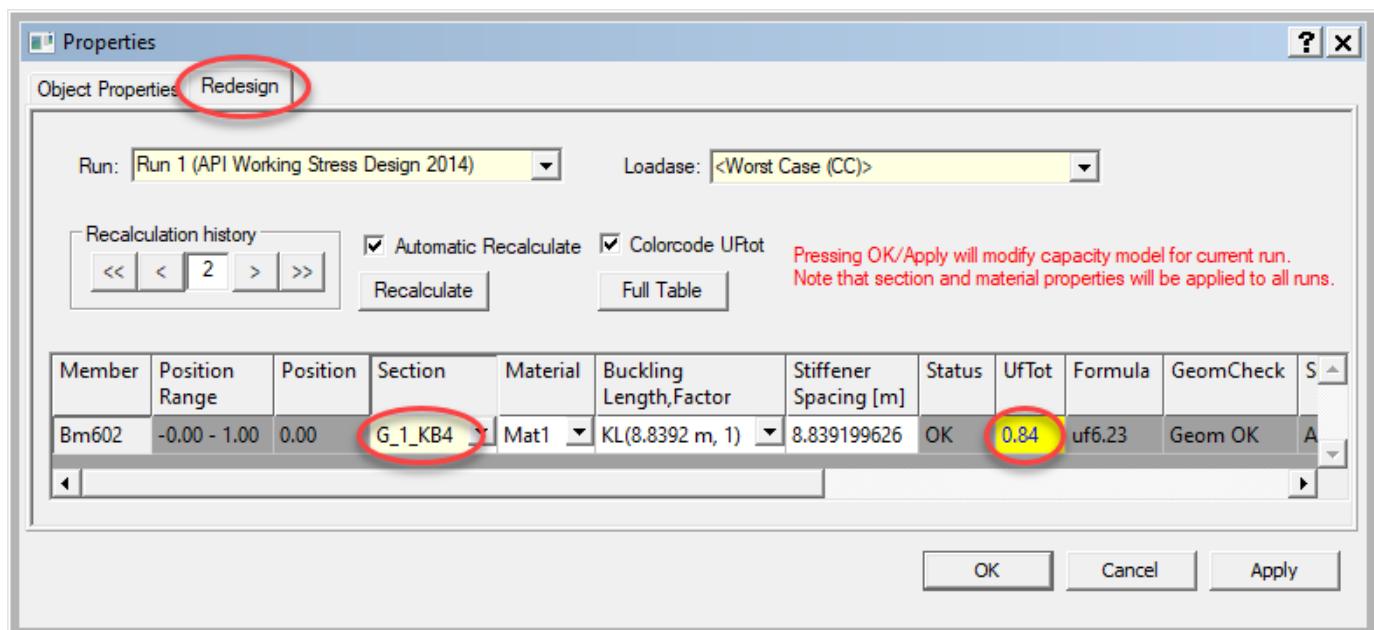


7 REDESIGN BY EDITING BEAM CROSS SECTION PROPERTIES

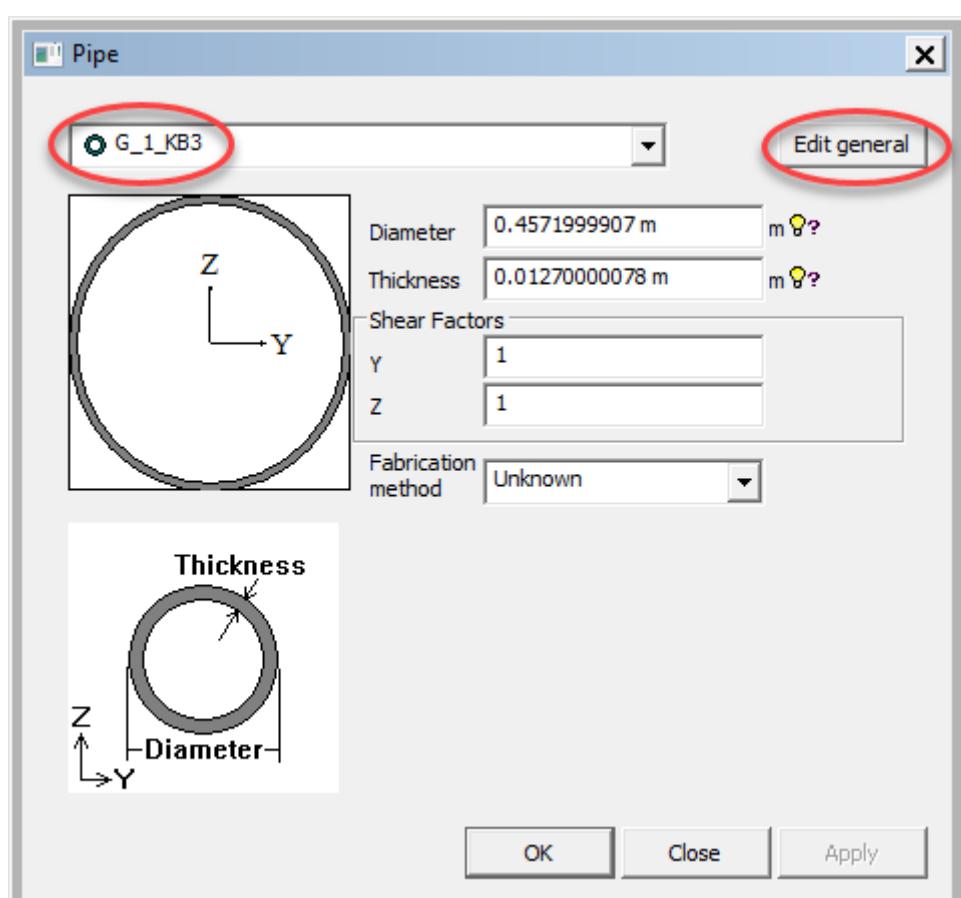
- Edit section properties by right-clicking a beam cross section in the browser, e.g. the one named W36X230, and selecting *Edit Section*.
- The strength of a section can be increased (but also its weight) by adding a fillet. A fillet of 25 mm has been added below.
- Click *Apply* to make the change take effect.
- Click *Edit general* to see derived section properties updated with the fillet.
- A next step could be to rerun the structural analysis and code checking by right-clicking the capacity manager and selecting *Run All*.



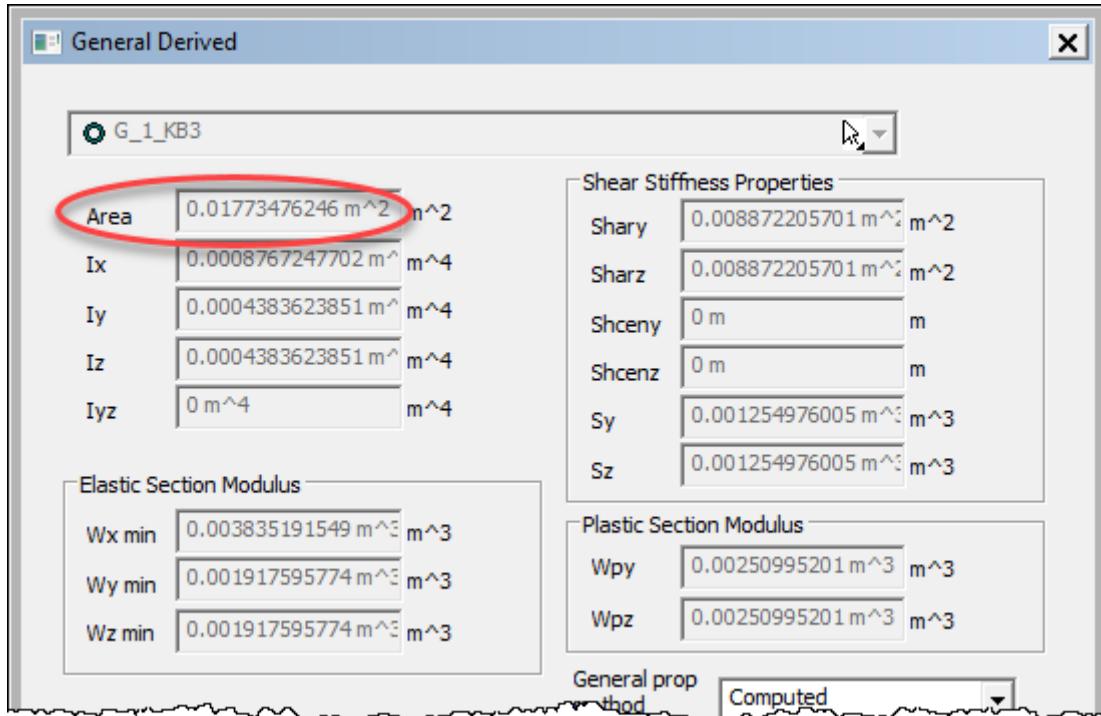
- In the code check above the member Bm602 with tubular section G_1_KB3 failed.
- Redesigning this member would normally involve right-clicking it and selecting *Redesign*. And in the *Redesign* tab of the *Properties* dialog, select a larger pipe section, e.g. G_1_KB4.



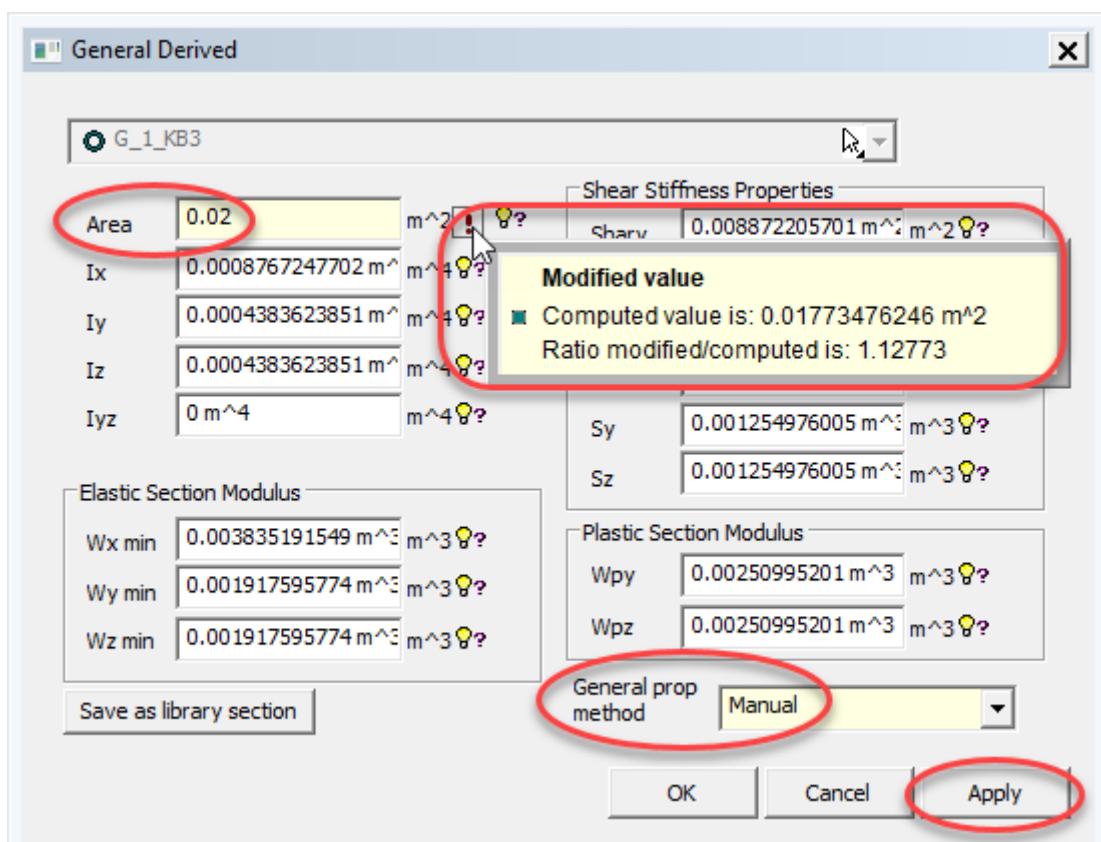
- But in this case, for illustration purposes, we shall edit the properties for section G_1_KB3 by increasing its area from 0.01773476 m^2 to 0.02 m^2 .
- Right-click G_1_KB3 found in the browser and select *Edit Section*.
- In the *Pipe* dialog click *Edit general*.



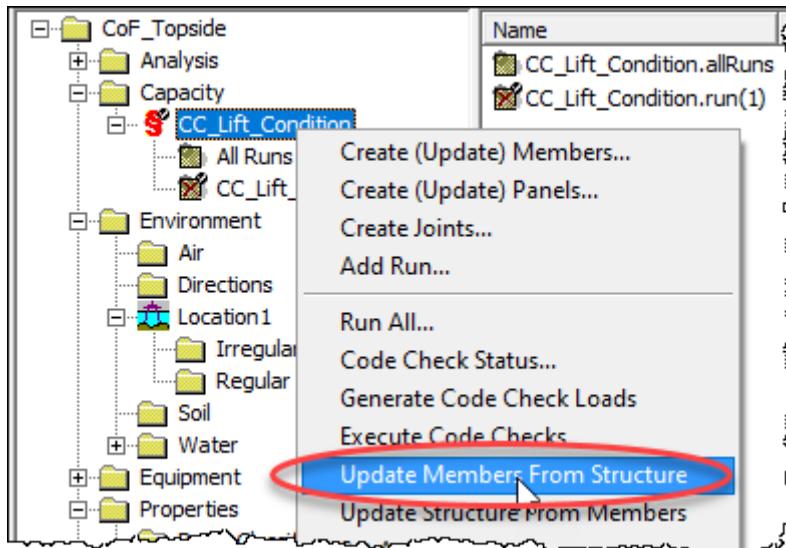
- The *General Derived* dialog shows the current area of 0.01773476 m^2 .



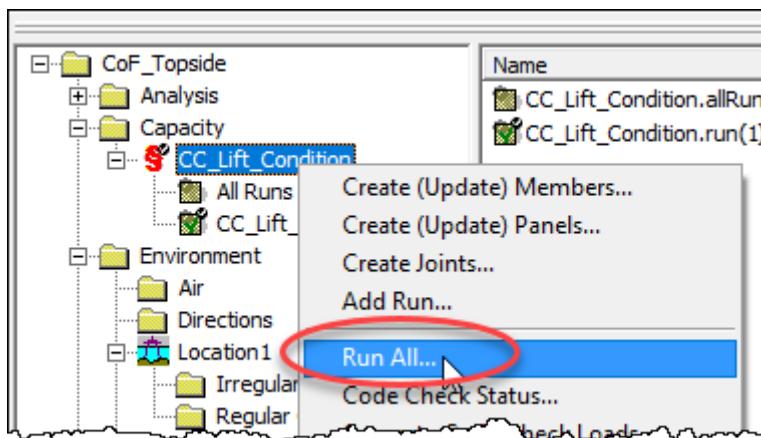
- To be allowed to modify derived values switch *General prop method* from *Computed* to *Manual*. Enter 0.02 for *Area*. Notice the information provided when hovering the exclamation mark for *Area*. Click *OK* or *Apply* to make the change take effect.



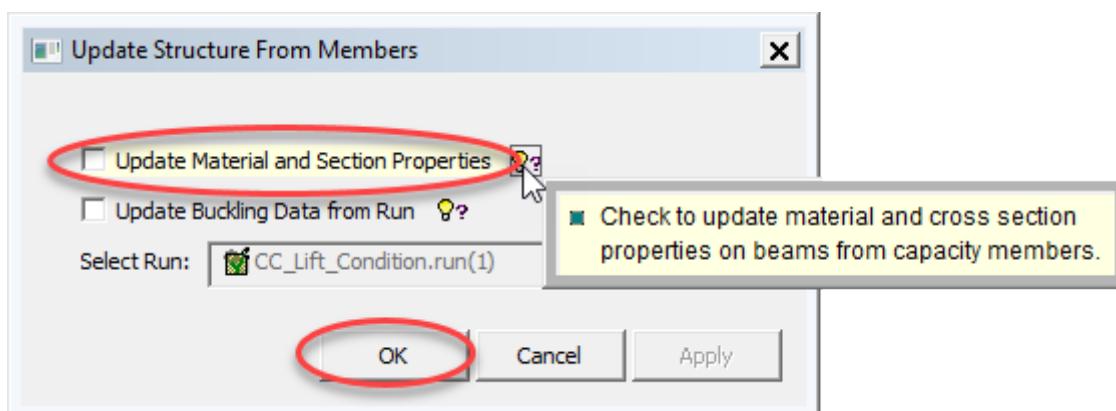
- To transfer the modified section from the concept model (structure) to the capacity members, right-click the capacity manager and select *Update Members From Structure*.



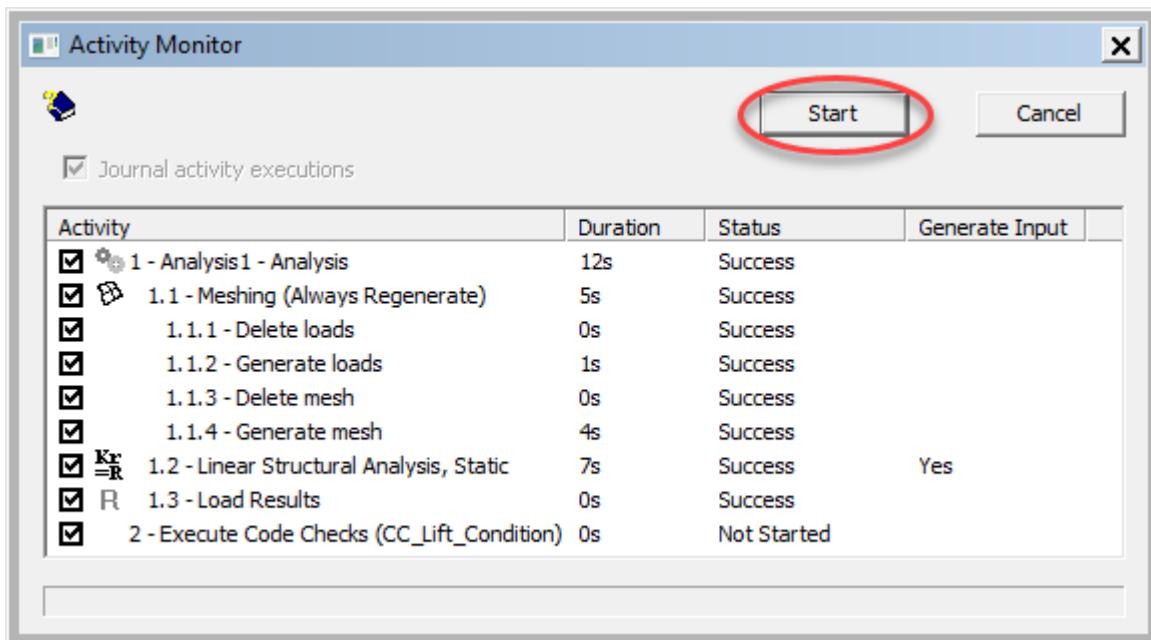
- Then right-click the capacity manager again and select *Run All*.



- The *Update Structure From Members* dialog appears. For consistency, uncheck *Update Material and Section Properties* to *not* transfer changes that might have been done to the capacity members back to the concept model (structure). But in this case no such changes have been done. Click *OK* to close the dialog.



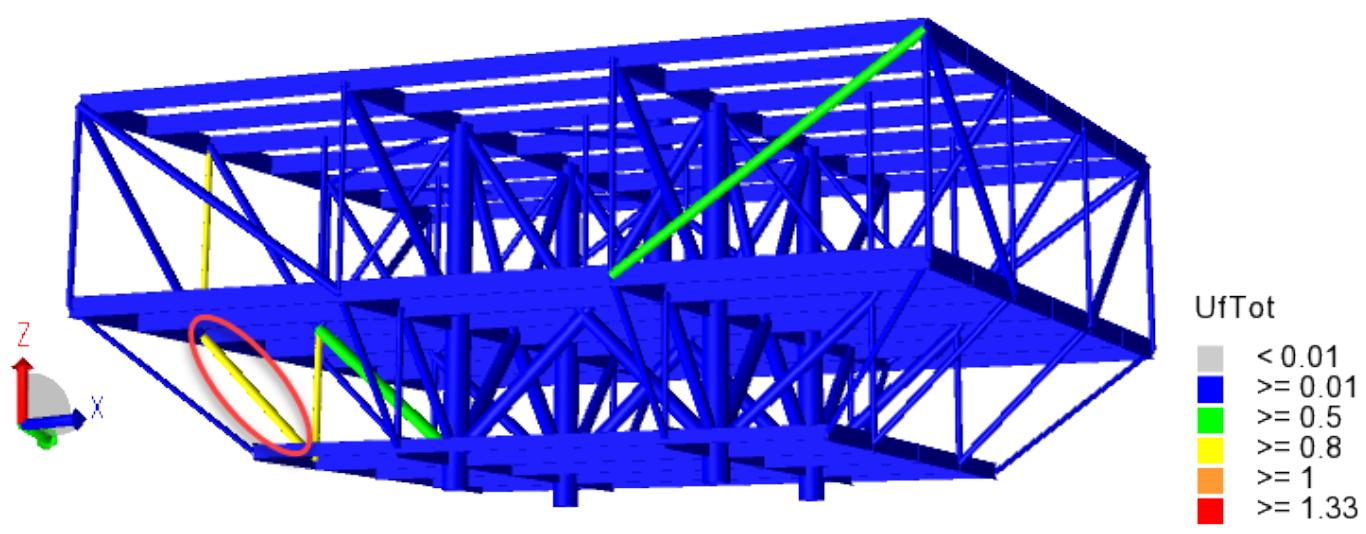
- The *Activity Monitor* appears. Click *Start* to run both the structural analysis and the code check.



- The utilisation of member Bm602 has been reduced to 0.97:

Capacity Model	LoadCase	Position	Status	UfTot	Formula	SubCheck
/ member(Bm602)	L2	0.00	OK	0.97	uf6.23	API WSD 2014 member
/ member(Bm568)	L2	1.00	OK	0.96	uf6.21	API WSD 2014 member
/ member(Bm563)	L2	0.00	OK	0.81	uf6.21	API WSD 2014 member

- The code check results shown graphically, the failing member Bm602 is now yellow:



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.