SESAM TUTORIAL

GeniE

Import SACS Topside Model

Valid from program version 8.2

Sesam Tutorial

GeniE – Import SACS Topside Model

Date: June 2021

Valid from GeniE version 8.2

Prepared by: Digital Solutions at DNV

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

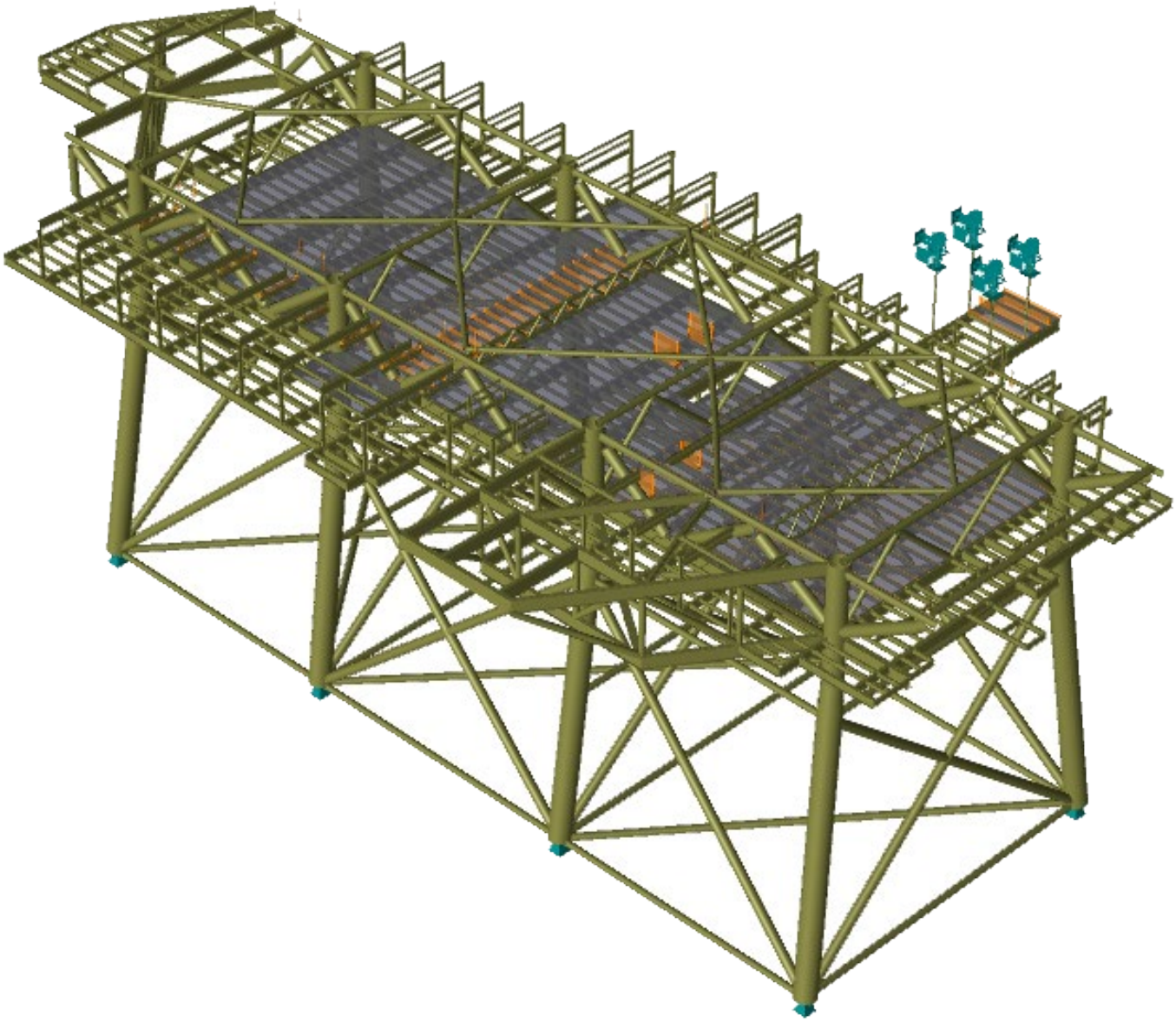
E-mail sales: digital@dnv.com

TABLE OF CONTENTS

1. Introduction Page 4
2. Open New Workspace Page 5
3. Import SACS Topside Model Page 6
4. Verify Imported Model Page 7
5. Check Imported Beam Properties Page 9
6. Check Imported Wishbones (Leg-Pile Connections) Page 10
7. Connect Top of Legs with Piles Page 12
8. Check Imported Support Points (Boundary Conditions) Page 14
9. Sets of the Imported Model Page 15
10. Load Cases and Load Combinations of the Imported Model Page 16
11. Create and Assign Mesh Density Page 17
12. Run Static Analysis Page 18
13. Present Analysis Results Page 19
14. Code Checking Page 20

1 INTRODUCTION

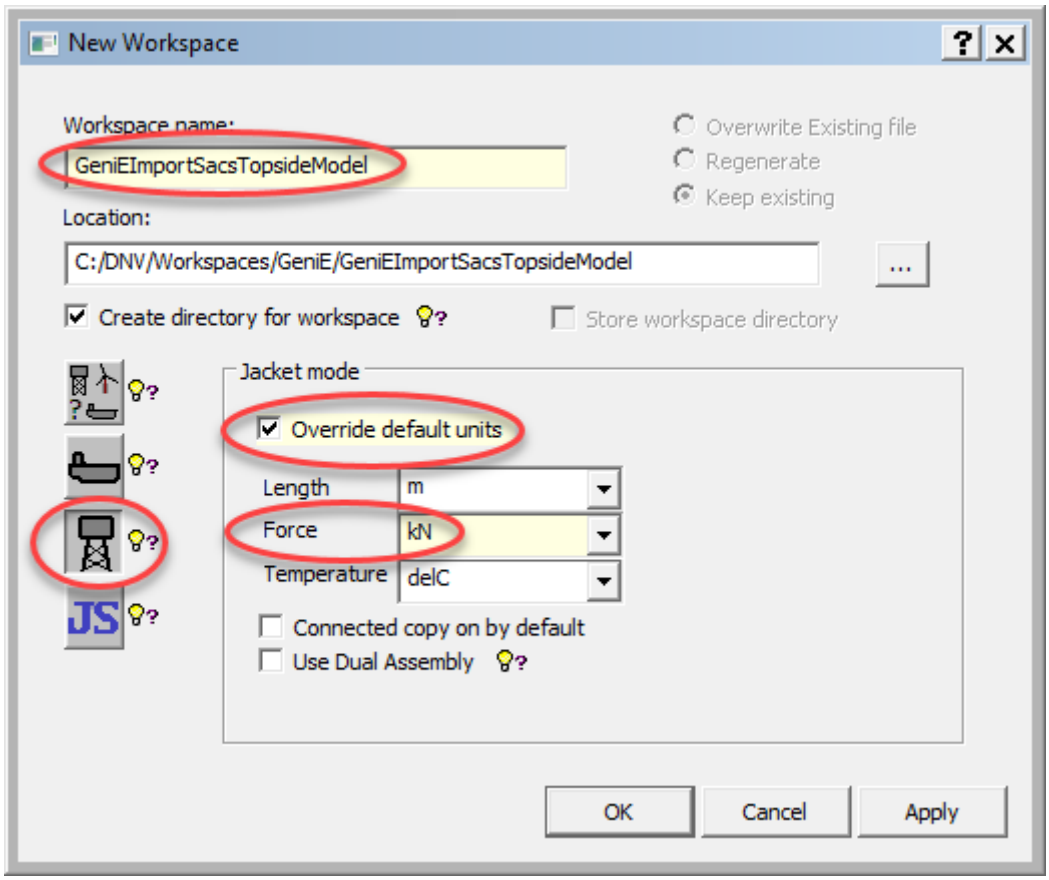
- This tutorial explains how to import a SACS topside model into GeniE.
- After importing the SACS model linear static analysis and code checking is performed.
- The SACS model includes:
  - Beams and shells
  - Joints
  - Supports
  - Loads
- This tutorial assumes basic knowledge in GeniE as covered by tutorial B1.



2 OPEN NEW WORKSPACE

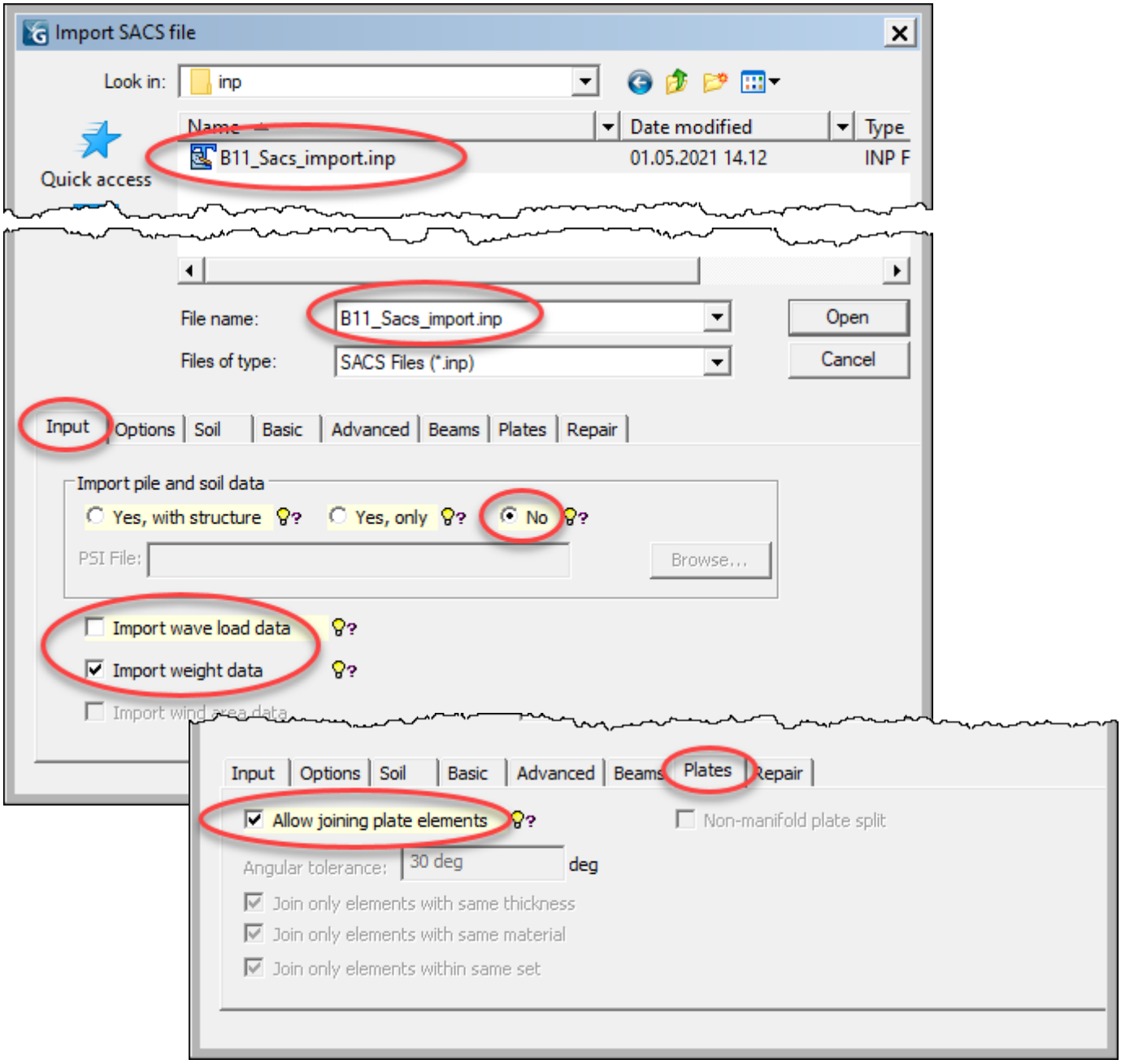
➢ Start GeniE and open a new workspace.

- Give a *Workspace name*.
- Check *Override default units* and change *Force* to *kN*.
- 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.



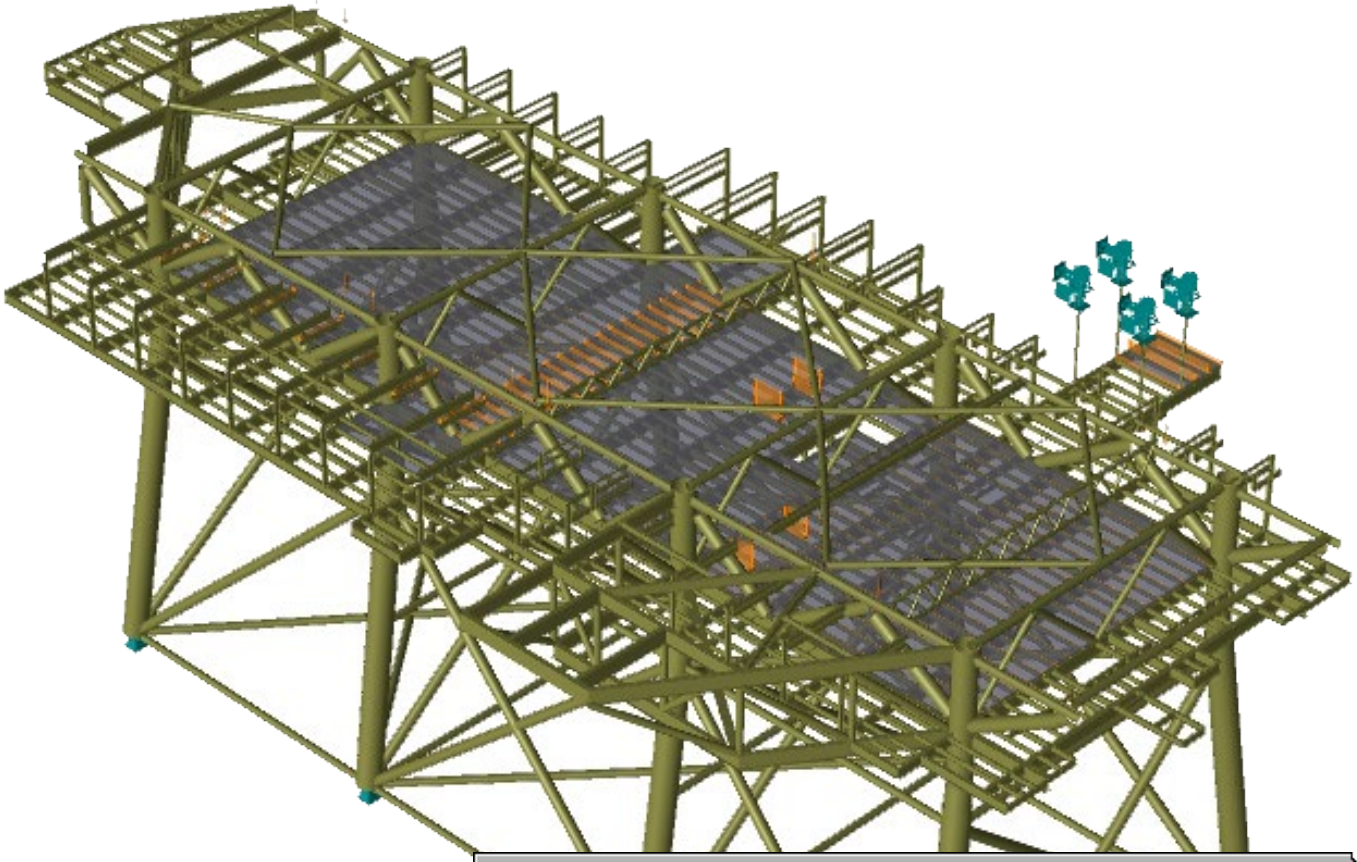
3 IMPORT SACS TOPSIDE MODEL

- Import the SACS model by *File* | *Import* | *SACS file*.
- Find the SACS input file B11_Sacs_import.inp in the installation folder: <path>\GeniE VX.Y-ZZ\Help\Tutorials\TutorialsBasicAndCodechecking\ B11_SACS_import\inp
- In the *Input* tab set *Import pile and soil data* to *No*. There are no wave loads either, so uncheck *Import wave load data*. Keep *Import weight data*.
- In the *Plates* tab, check *Allow joining plate elements*. This involves that coplanar plates in the SACS model are joined to single plate concepts.
- Accept default options in the other tabs and click *Open* to execute the import.

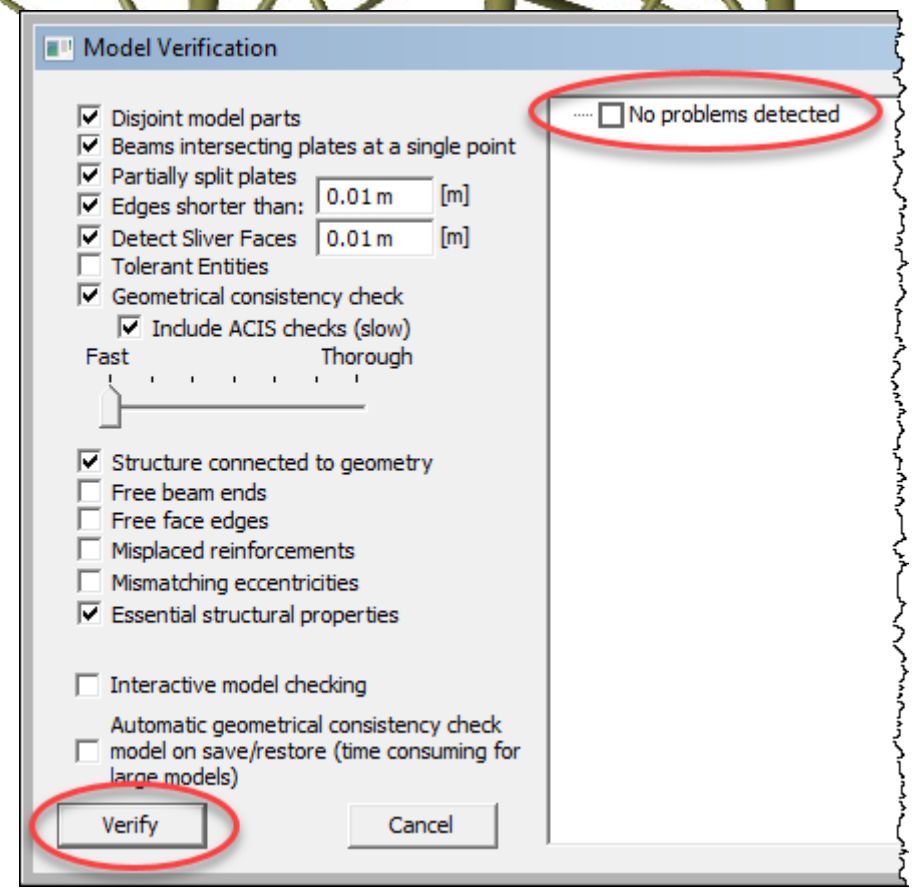


4 VERIFY IMPORTED MODEL

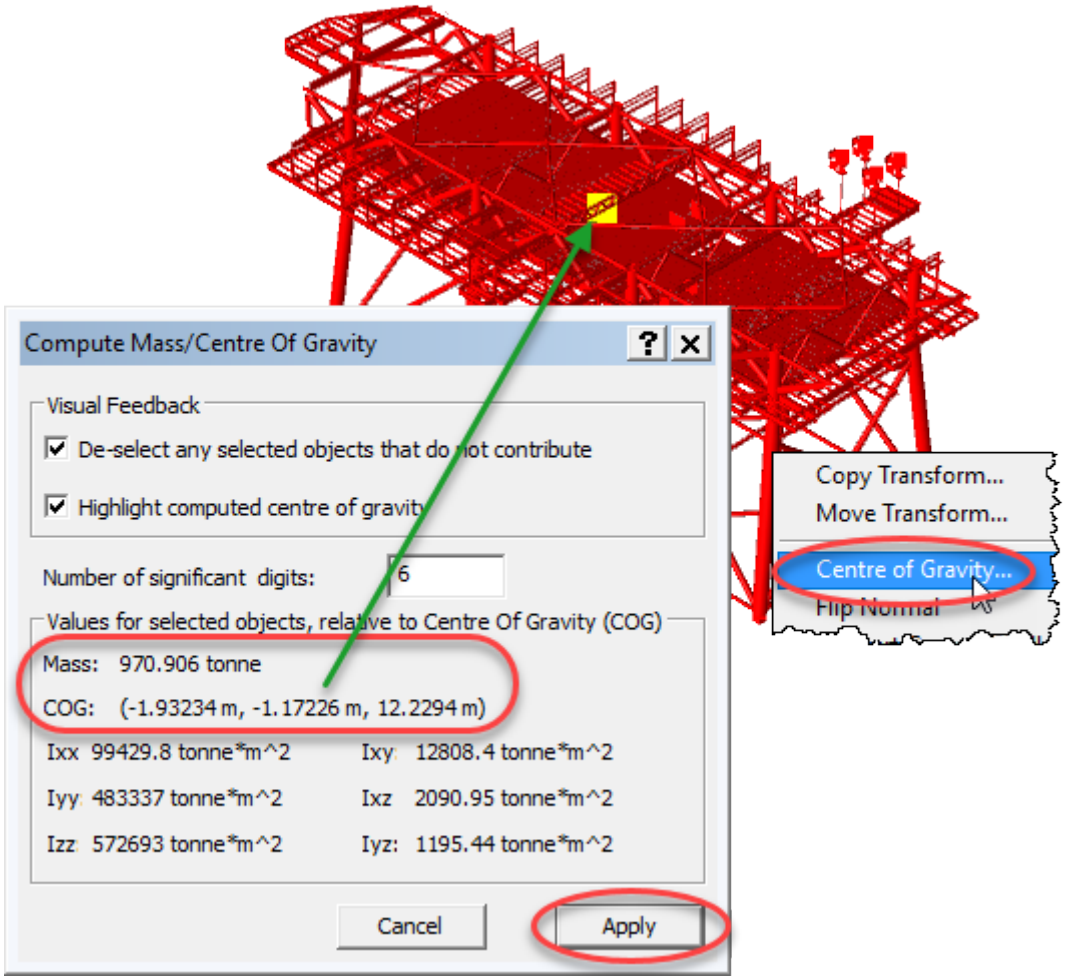
- The imported model should appear as shown below.



- Verify the consistency of the model by *Structure* | *Topology* | *Verify Model*. Accept default settings in the dialog, click *Verify* and see that no problems are detected.



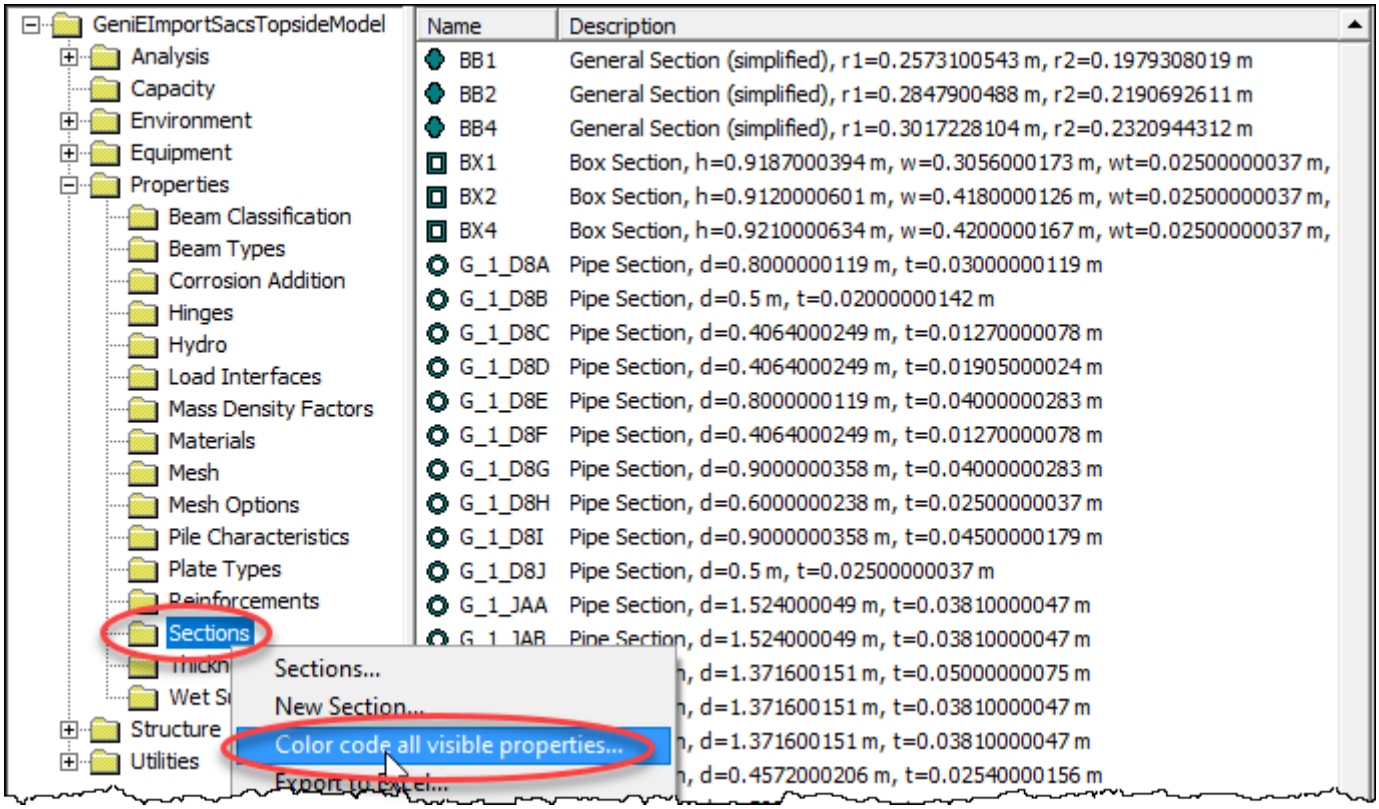
- Verify the total mass and COG of the imported model by dragging a rubberband to select the whole model, right-click and select *Centre of Gravity*. In the dialog appearing click *Apply* to find the mass and COG. The COG is also marked by a yellow square. See below.

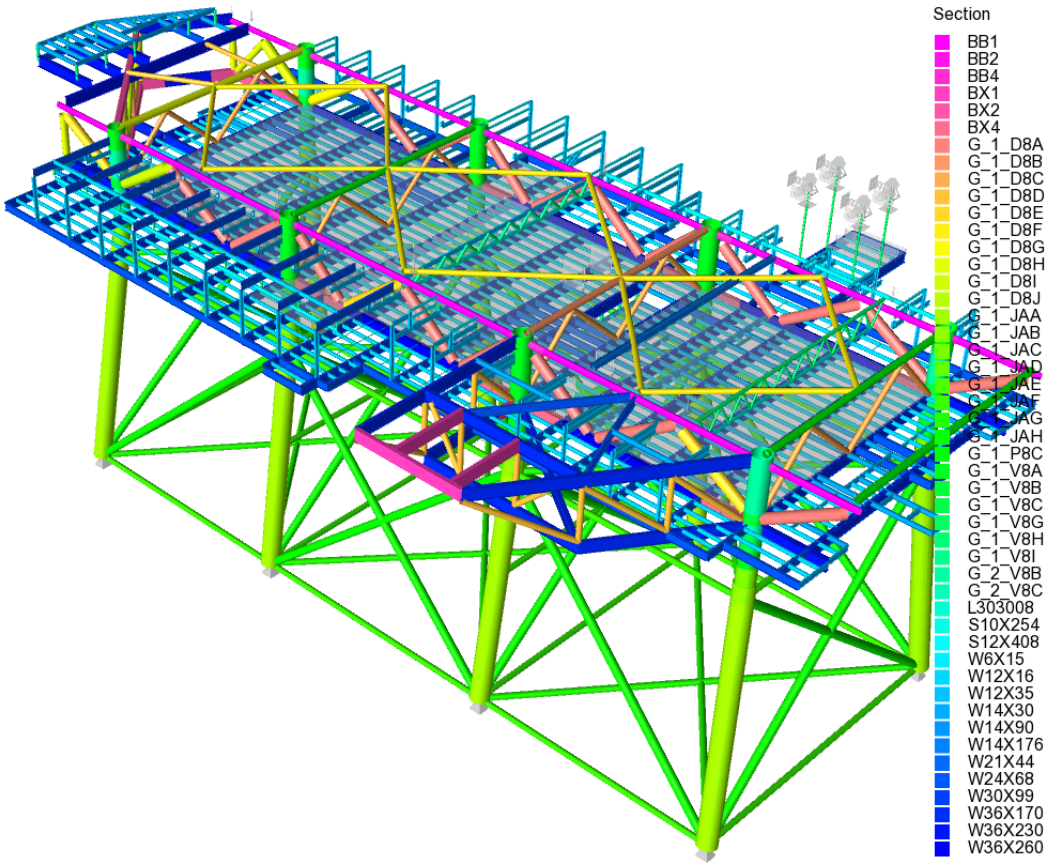


- The mass and COG should be compared with data from the SACS model.

5 CHECK IMPORTED BEAM PROPERTIES

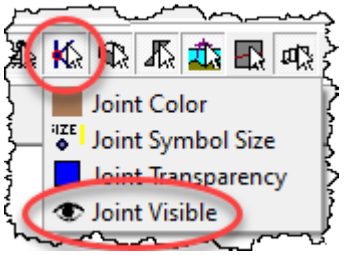
➢ Open the *Properties* | *Sections* folder to see the imported beam cross sections. Right-click the folder and select *Color code all visible properties* to colour code the beam cross section used in the model.

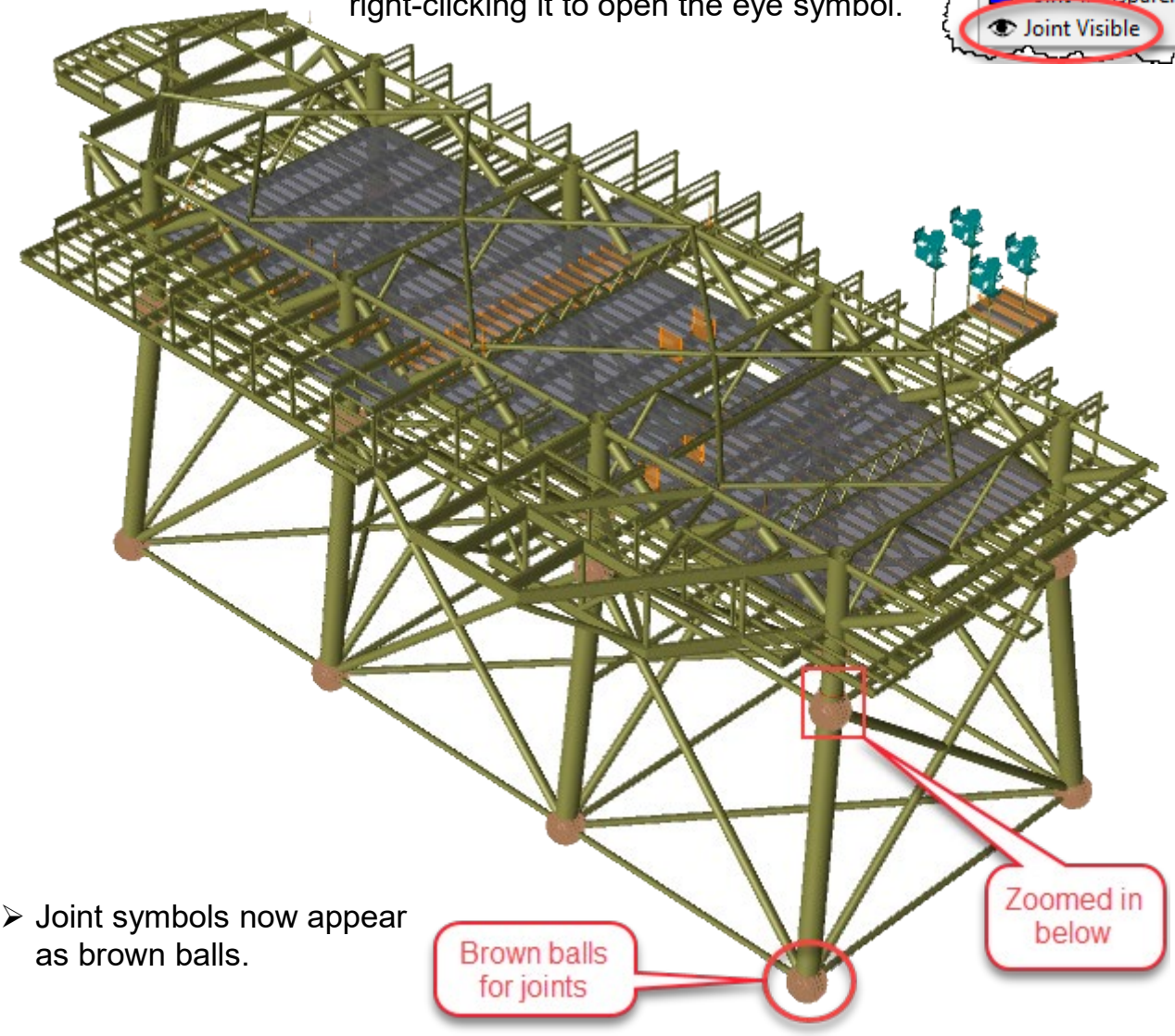




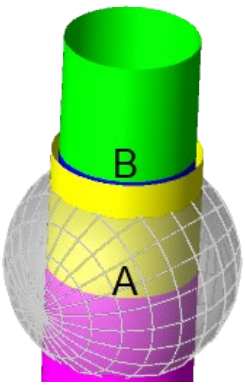
6 CHECK IMPORTED WISHBONES (LEG-PILE CONNECTIONS)

- The SACS model contains joints but these were not imported. (To import joints check *Import named joints* in the *Options* tab of the *Import SACS file* dialog.)
- But GeniE creates joints where there are so-called wishbones in the SACS model. Wishbones in SACS are leg-pile connections. To display these joints created by GeniE and allow them to be selected press *Joint selection* down and right-clicking it to open the eye symbol.

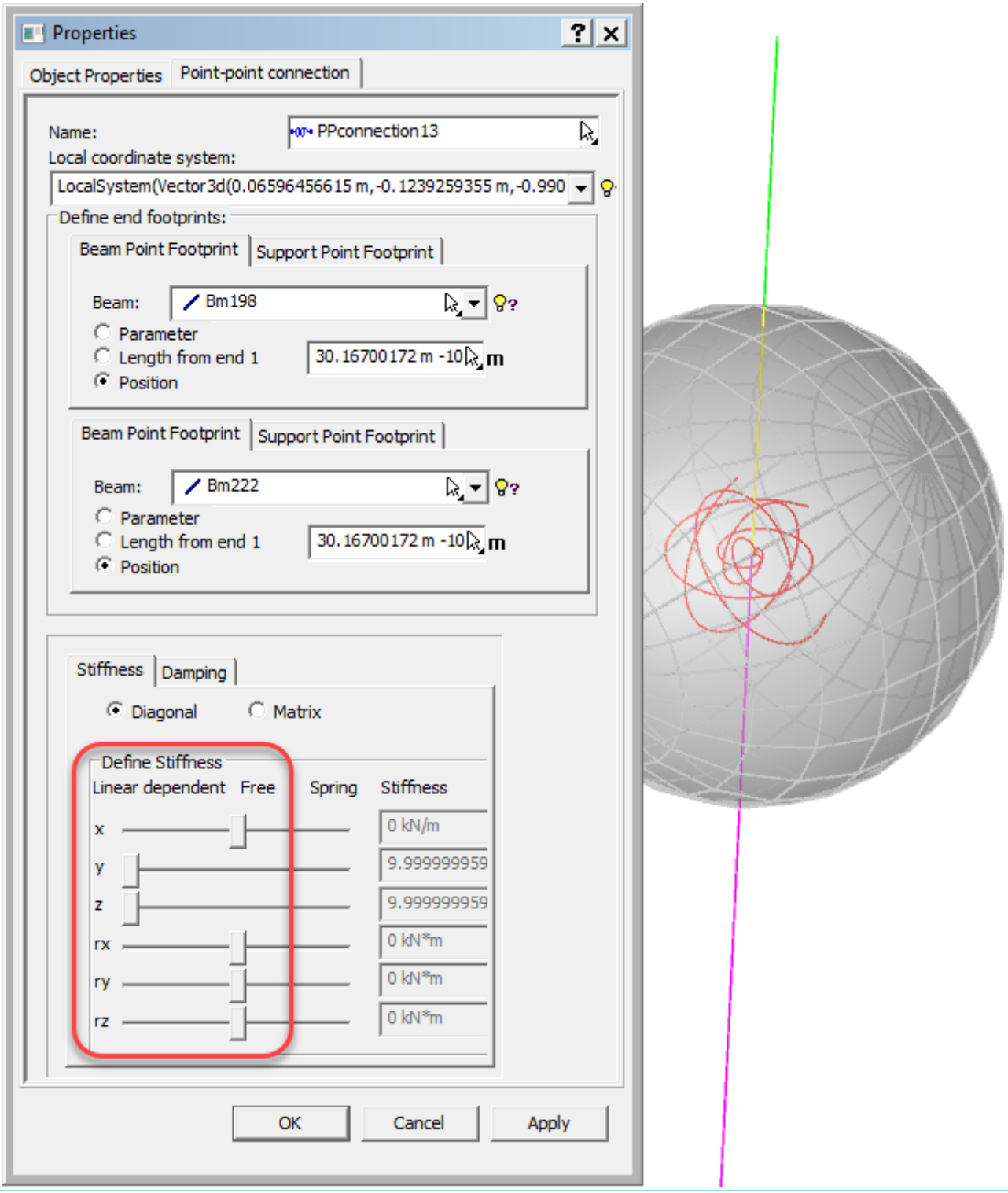




- Joint symbols now appear as brown balls.
- Understand this model further:
  - Select a leg, pile and corresponding joint. Zoom in on the leg top. Colour code cross sections (*Color code all visible properties*).
  - A joint is positioned at the transition between magenta and yellow, point A in the figure. This is also where bracings connect.
  - The pile (blue, barely visible, and green) is disconnected from the leg (yellow) at point B. This is incorrect and will be dealt with later.

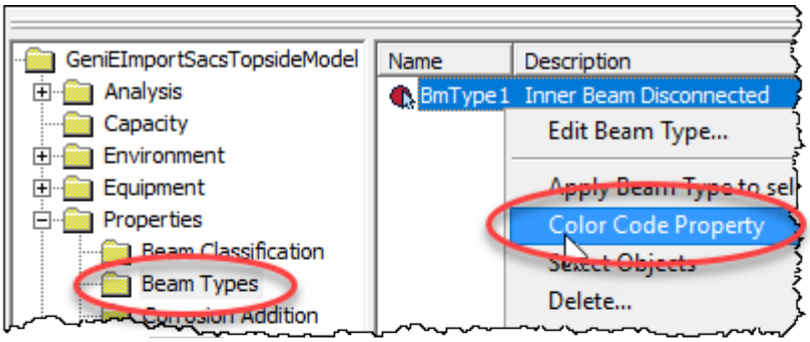


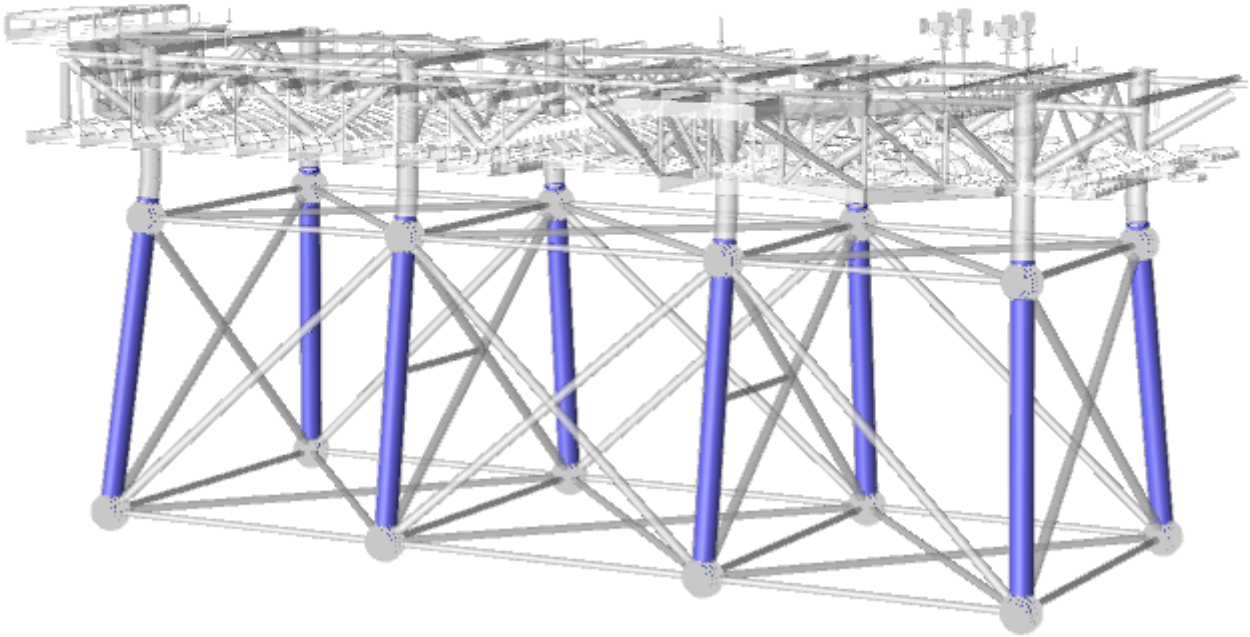
- Switch to wireframe view to see the point-point connection displayed as coils inside the joint.
- Select the point-point connection and right-click to open the *Properties* dialog. Remove the joint from the display to do so. (Select the joint and press Alt+minus.)
- See that the leg and pile are connected only in the local transverse y and z directions. Longitudinally (x) and rotationally they are disconnected (free).



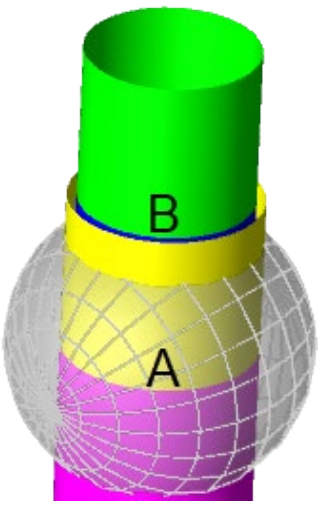
7 CONNECT TOP OF LEGS WITH PILES

- The import involves that all piles are assigned beam type *Inner Beam Disconnected*. This can be seen by colour coding this property as shown.

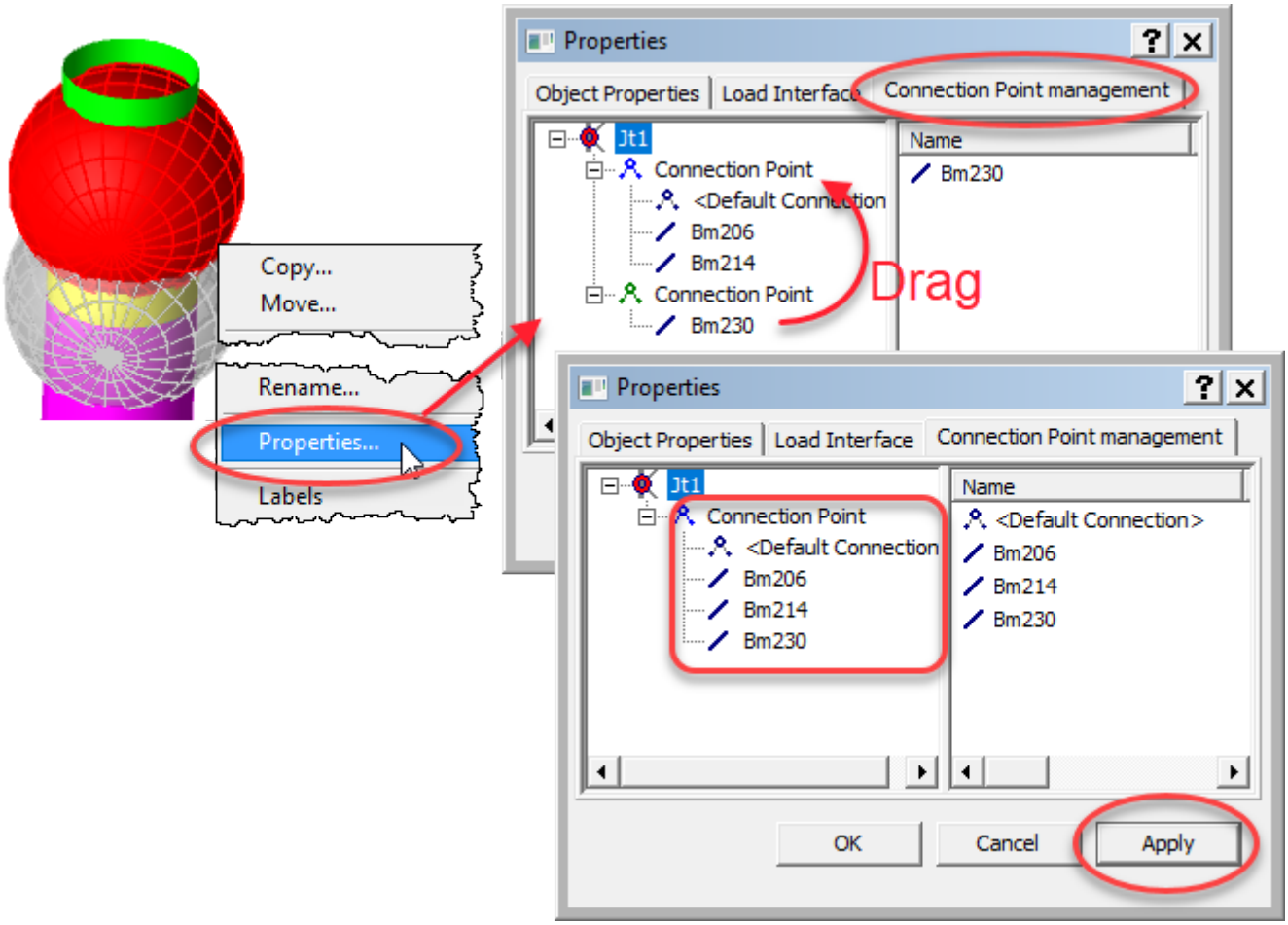




- This disconnection means that the pile (blue, barely visible, and green) is disconnected from the leg (yellow and magenta) at B in the figure below. This is incorrect and must be fixed as explained on the following page.



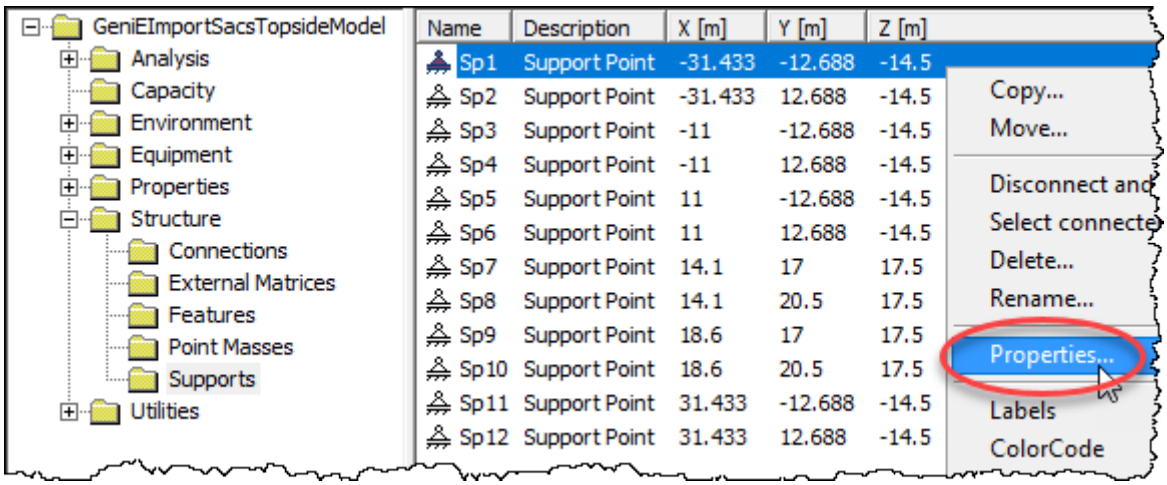
- First, introduce a joint at the top of the leg.
- Then open the *Properties* dialog and see that the joint contains two so-called *Connection Points*. This involves that the meshing will create two nodes, one belonging to the pile and the other belonging to the leg. I.e. the pile and leg will not be connected.
- Drag the beam (actually, the leg) of the lower *Connection Point* into the upper (and default) *Connection Point* and click *Apply*. See that the joint now contains a single *Connection Point*. This means that the meshing will create a single node to which all beam elements are connected.

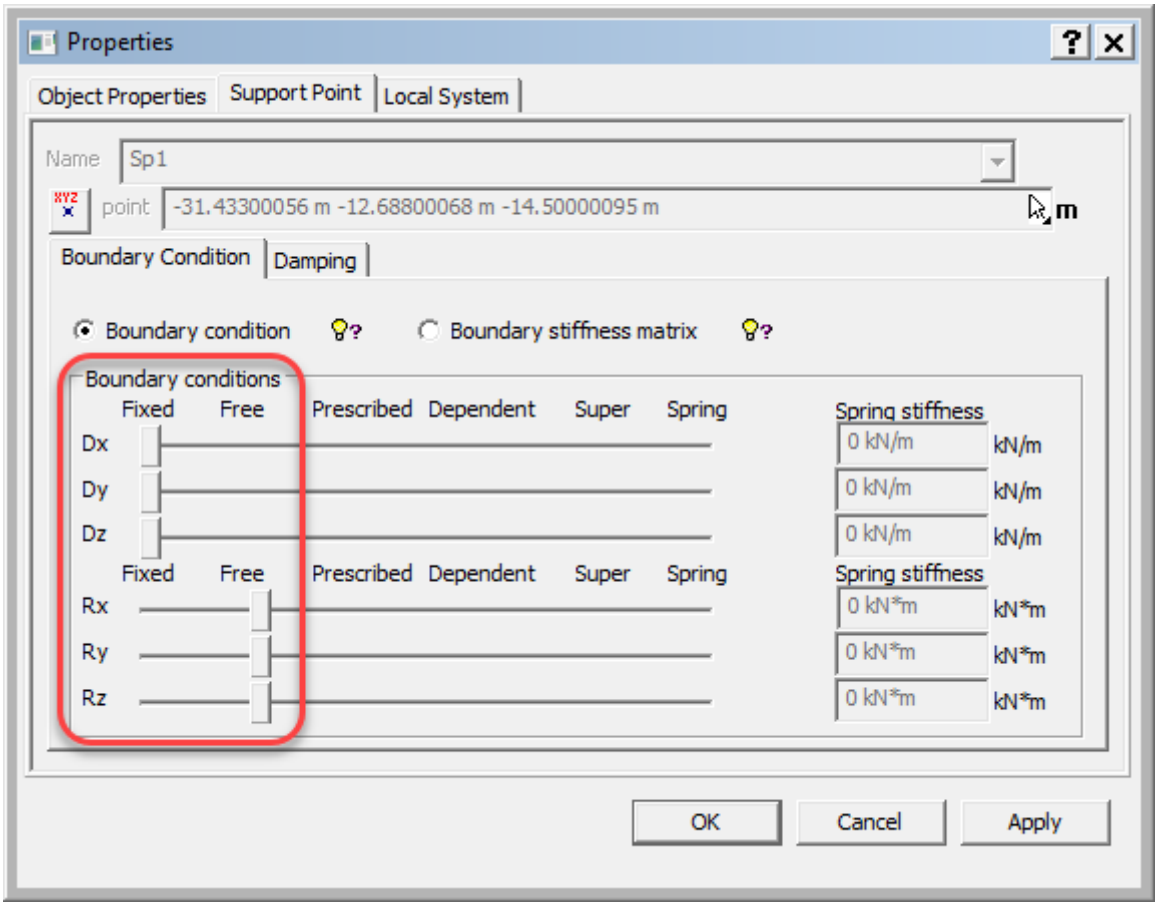


- Do this for all eight legs.

8 CHECK IMPORTED SUPPORT POINTS (BOUNDARY CONDITIONS)

- Find all support points in the *Structures* | *Supports* folder. See that the eight legs are pinned at their lower points. Be aware of that the support points may be hidden inside the joints. Remove the joints from the display to see the support points.

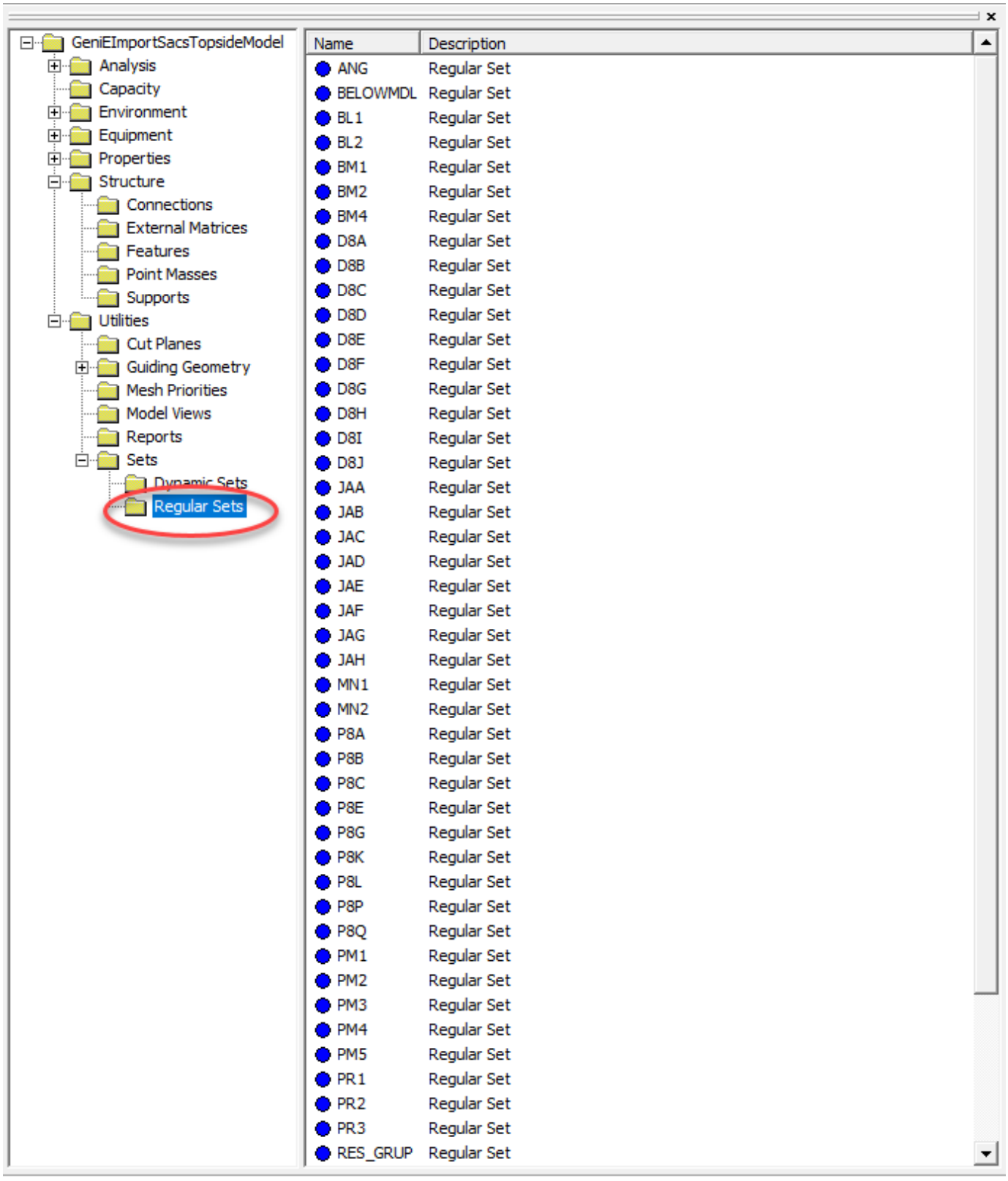




- In addition, four points at the deck are supported in the vertical direction by spring-to-grounds. These represent a bridge connection to a neighbouring platform.

9 SETS OF THE IMPORTED MODEL

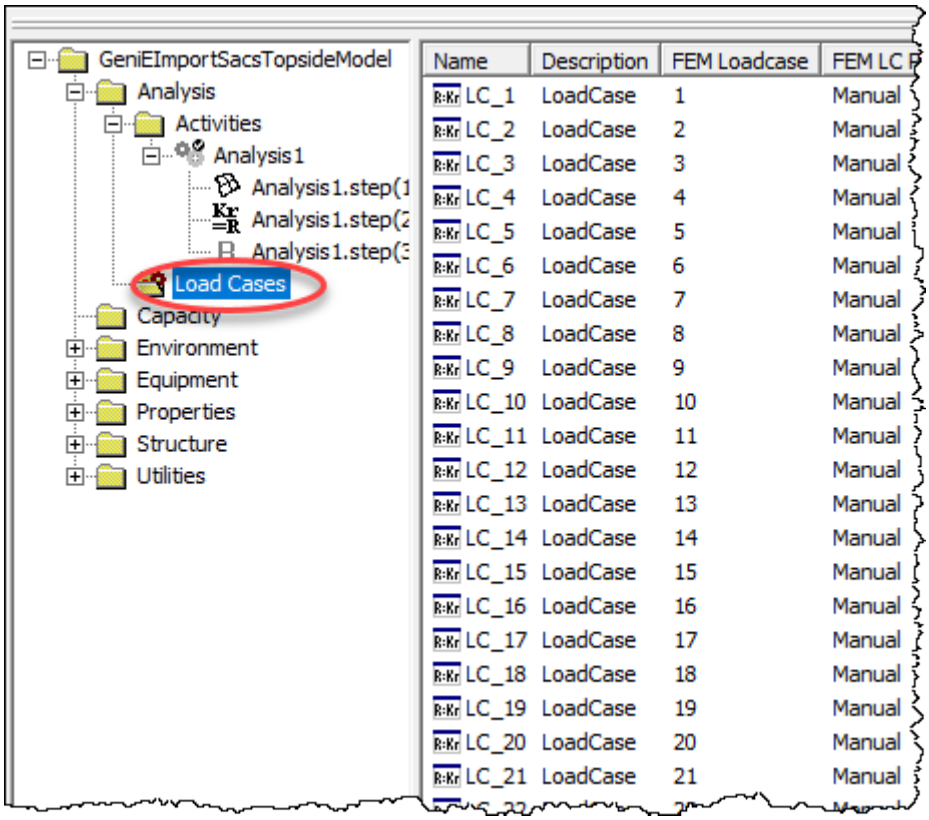
➢ Find the sets of the imported model in folder *Utilities* | *Sets* | *Regular Sets*.



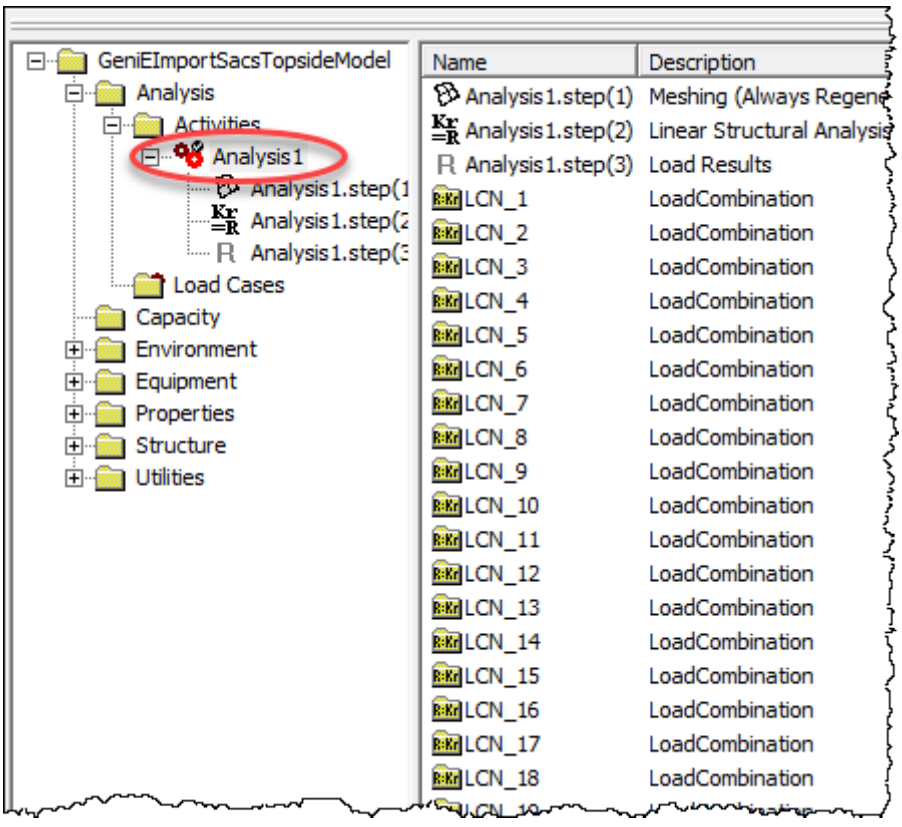
➢ Click sets and see that parts of the model are highlighted in red, i.e. selected.

10 LOAD CASES AND LOAD COMBINATIONS OF THE IMPORTED MODEL

- Find 37 load cases in the *Analysis* | *Load Cases* folder.



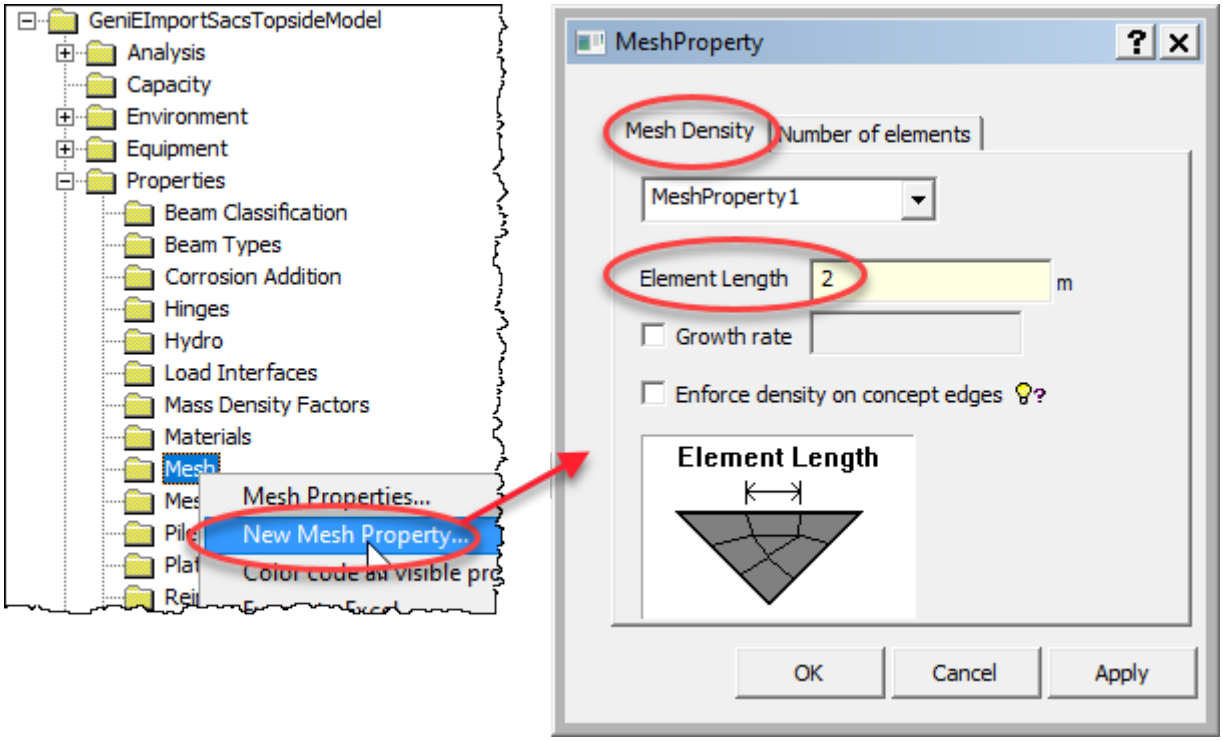
- Select the imported analysis activity named Analysis1 and find that there are 54 load combinations.
- In this case, the first 37 combinations are one-to-one with the 37 load cases, i.e. LCN_*i* contains only LC_*i* with factor 1. This is an intentional design of the importer to cater for cases in which wave and buoyancy loads are included in the basic load cases in the SACS model. This in turn relates to a GeniE requirement.
- The real combinations are named LCOM_*i*.



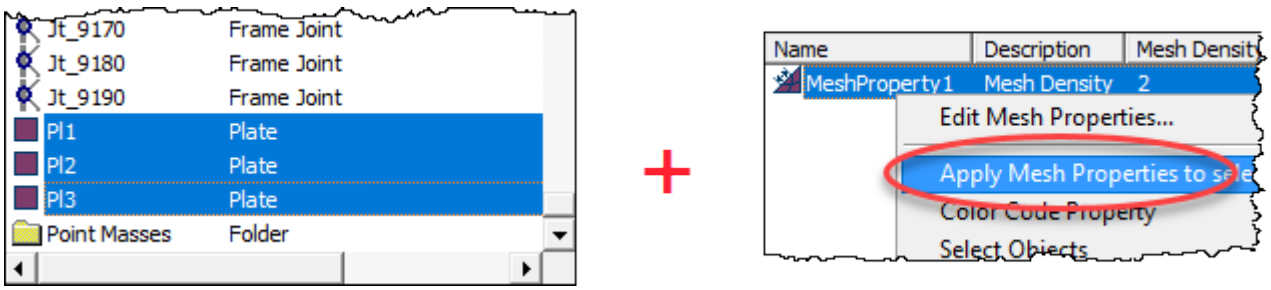
- For each load combination a property dialog can be opened to see which loads contribute to it.

11 CREATE AND ASSIGN MESH DENSITY

- Before running an analysis a mesh density must be defined and assign to the plates. Right-click the *Properties* | *Mesh* folder to create a *Mesh Density* with *Element Length* set to 2 m.

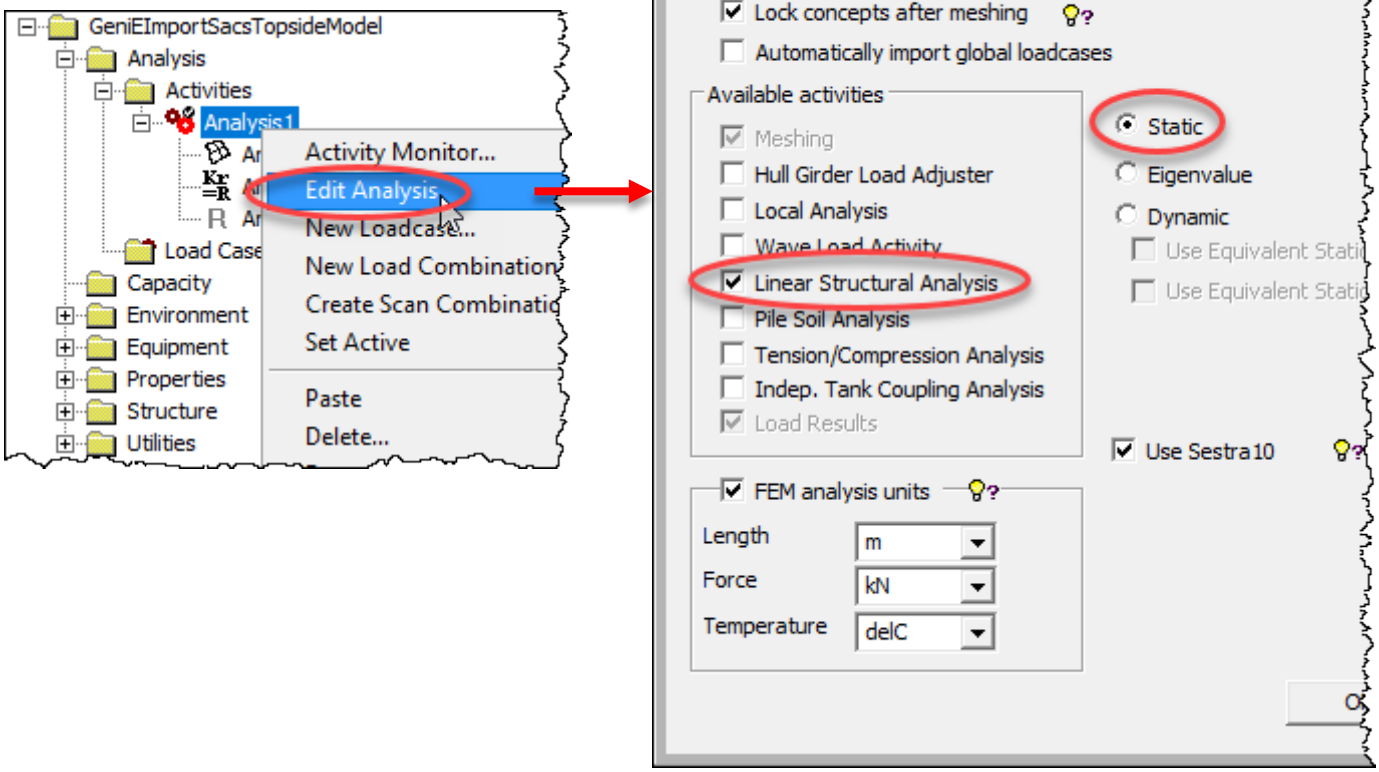


- Thereafter, assign (apply) this mesh property to all three plates. Find the plates in the *Structure* folder (at bottom when sorted on *Name*).

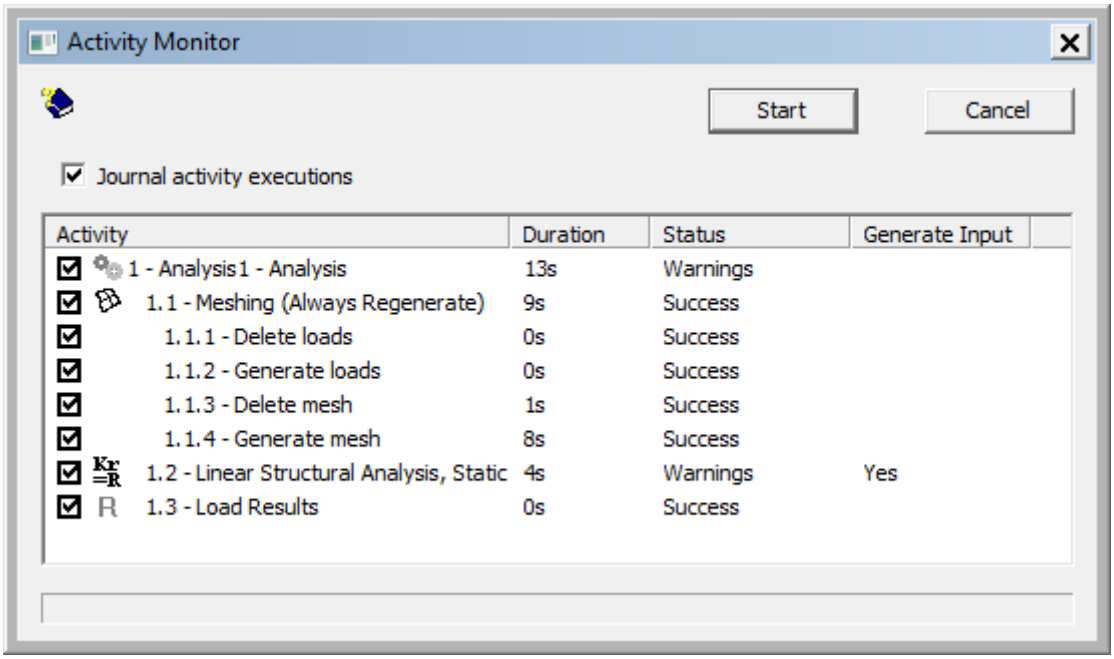


12 RUN STATIC ANALYSIS

- Right-click the imported analysis activity (Analysis1) to edit it. Verify that this is a linear static analysis.

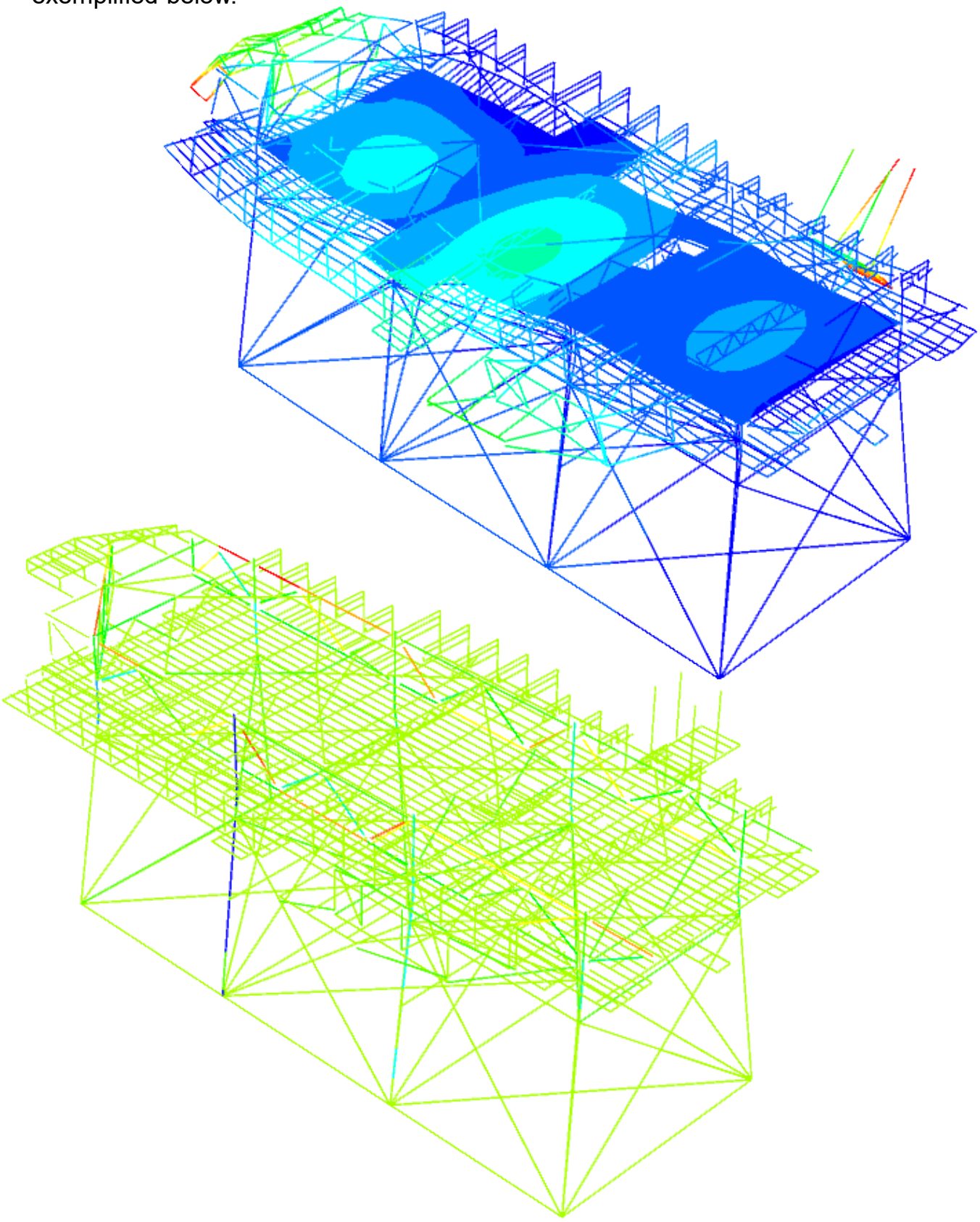


- Use *Mesh & Analysis* | *Activity Monitor* (or Alt+D) to open the *Activity Monitor* and click *Start* to run the analysis.
- The analysis in Sestra may get a *Warning Status*. This is likely due to a few bad element shapes and nothing to be concerned with in this tutorial.



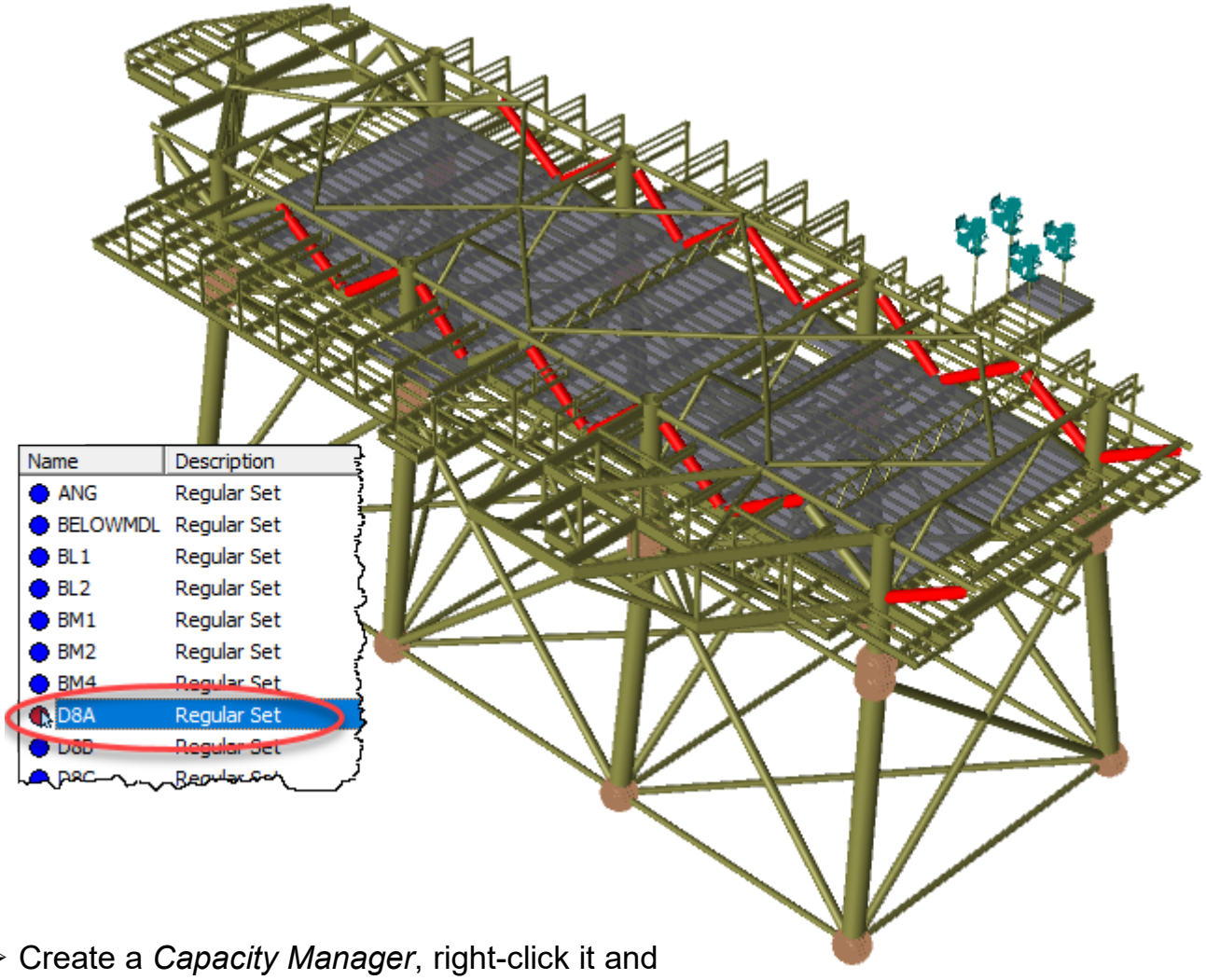
13 PRESENT ANALYSIS RESULTS

- Switch to *Results - All* display configuration, open the *Result presentation* dialog and present some results. E.g. deformed shape and beam axial forces as exemplified below.

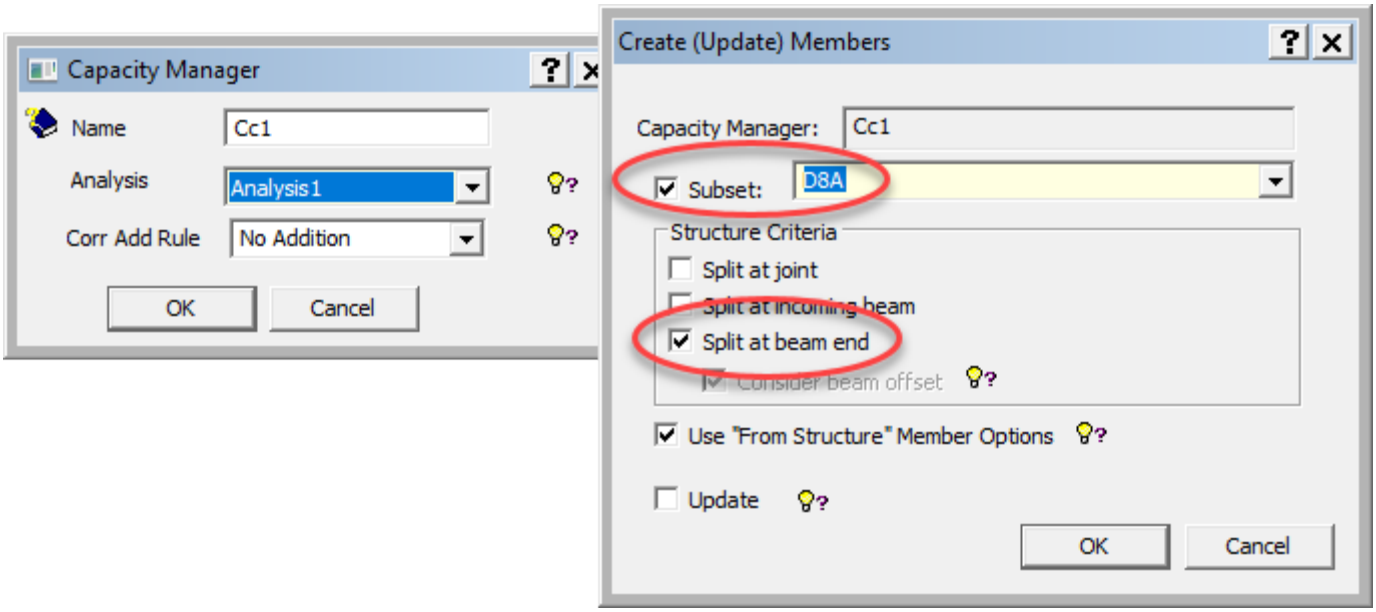


14 CODE CHECKING

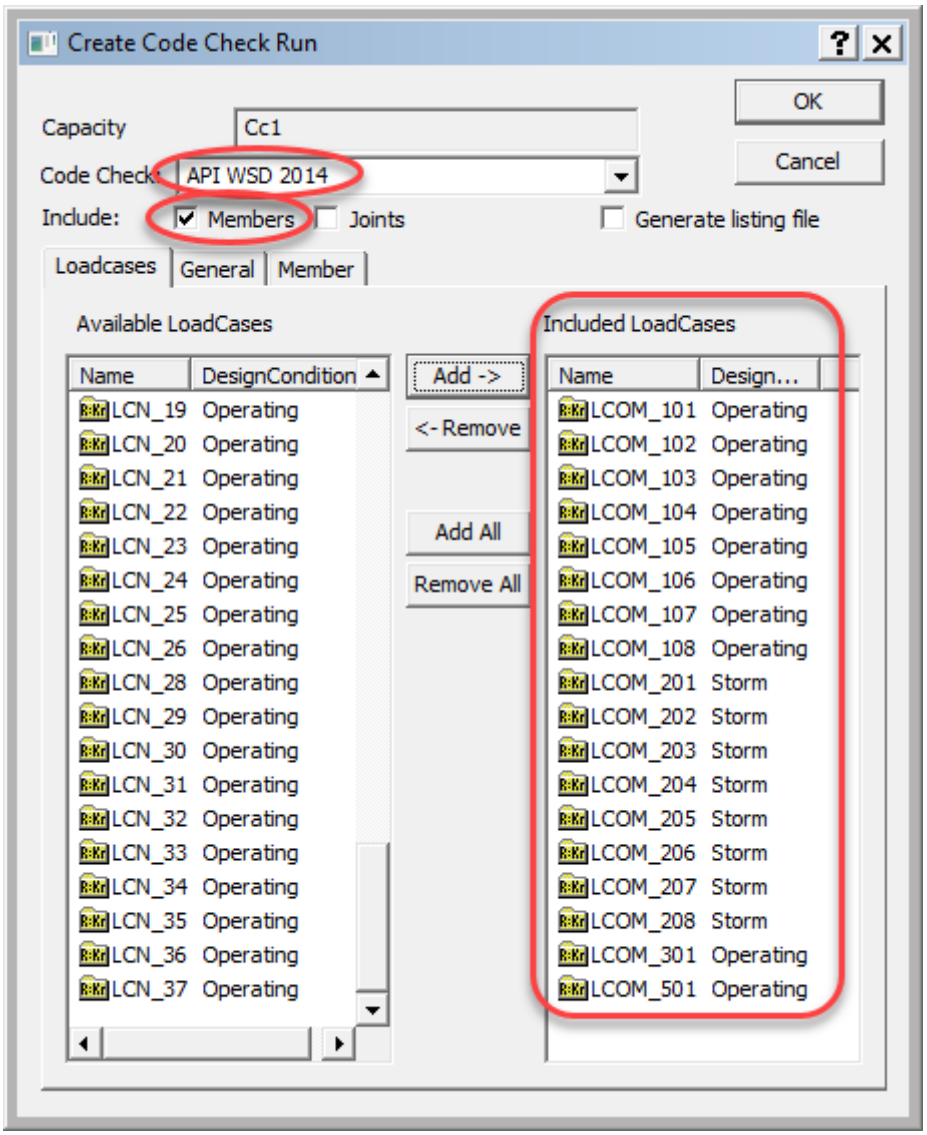
- Just to exemplify code checking, the set named D8A (imported from the SACS model) will be code checked according to API WSD 2014. The set is selected below.



- Create a *Capacity Manager*, right-click it and *Create (Update) Members* for it.

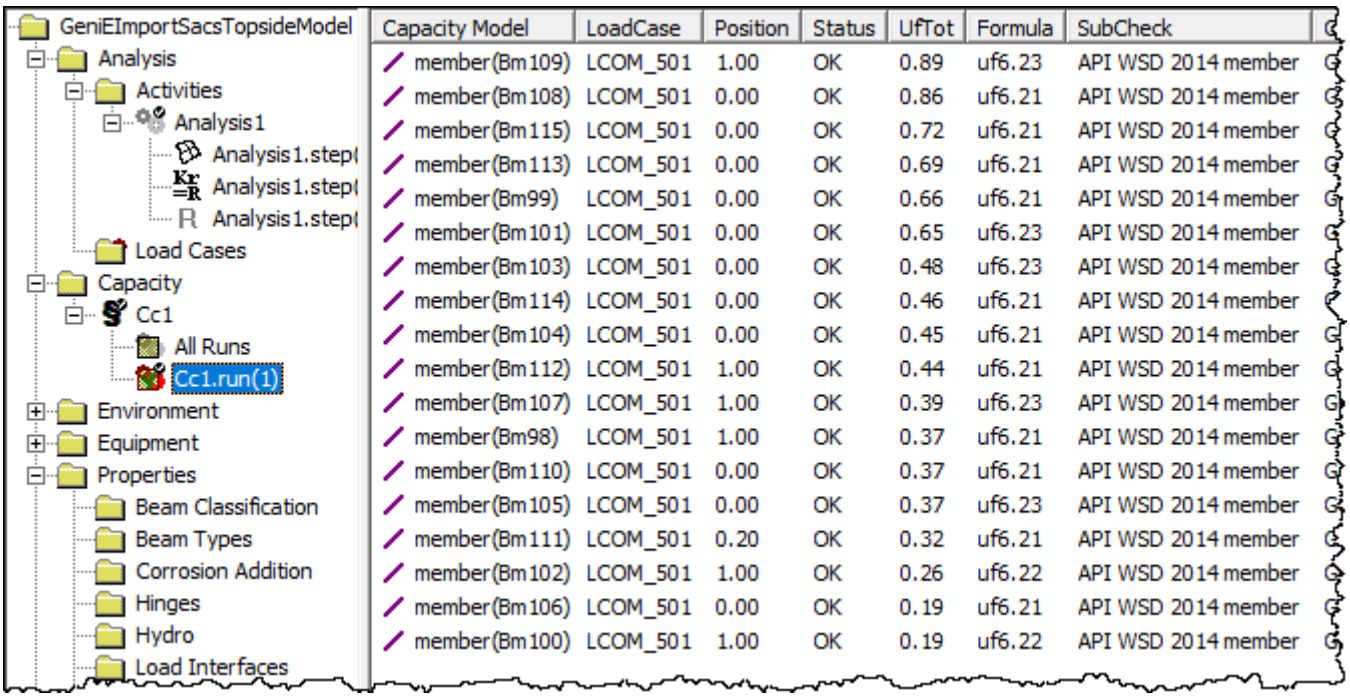


- Right-click the *Capacity Manager* once more to *Add Run*. In the dialog select *API WSD 2014*, *Members* (not *Joints*) and include all real combinations (named LCOM_*i*) for code checking.



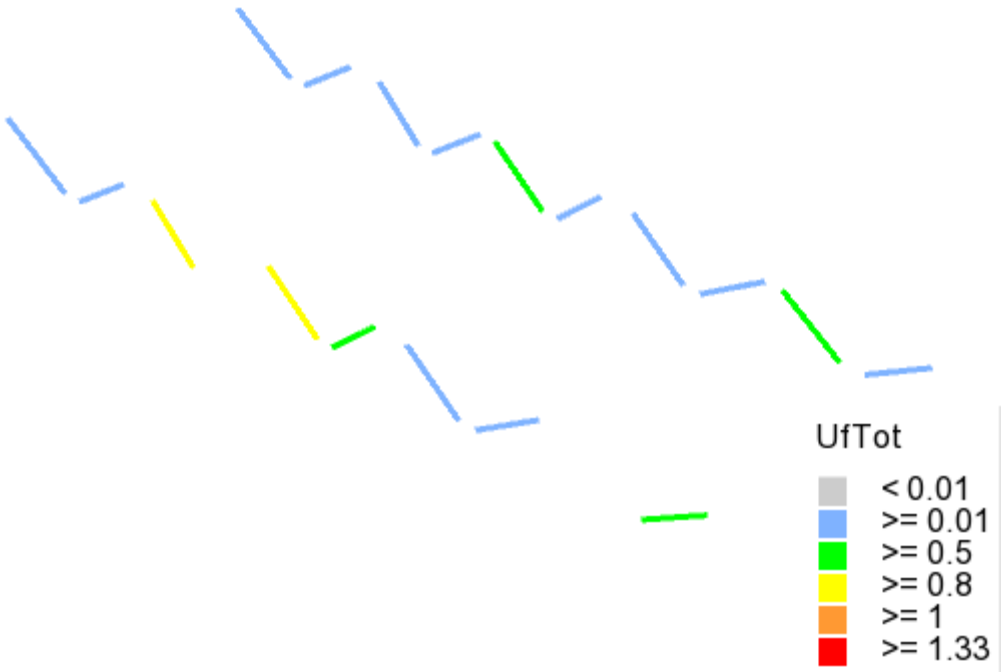
- Execute the code checking.

➢ The code check results are tabulated in the browser:



Capacity Model	LoadCase	Position	Status	UfTot	Formula	SubCheck
member(Bm109)	LCOM_501	1.00	OK	0.89	uf6.23	API WSD 2014 member
member(Bm108)	LCOM_501	0.00	OK	0.86	uf6.21	API WSD 2014 member
member(Bm115)	LCOM_501	0.00	OK	0.72	uf6.21	API WSD 2014 member
member(Bm113)	LCOM_501	0.00	OK	0.69	uf6.21	API WSD 2014 member
member(Bm99)	LCOM_501	0.00	OK	0.66	uf6.21	API WSD 2014 member
member(Bm101)	LCOM_501	0.00	OK	0.65	uf6.23	API WSD 2014 member
member(Bm103)	LCOM_501	0.00	OK	0.48	uf6.23	API WSD 2014 member
member(Bm114)	LCOM_501	0.00	OK	0.46	uf6.21	API WSD 2014 member
member(Bm104)	LCOM_501	0.00	OK	0.45	uf6.21	API WSD 2014 member
member(Bm112)	LCOM_501	1.00	OK	0.44	uf6.21	API WSD 2014 member
member(Bm107)	LCOM_501	1.00	OK	0.39	uf6.23	API WSD 2014 member
member(Bm98)	LCOM_501	1.00	OK	0.37	uf6.21	API WSD 2014 member
member(Bm110)	LCOM_501	0.00	OK	0.37	uf6.21	API WSD 2014 member
member(Bm105)	LCOM_501	0.00	OK	0.37	uf6.23	API WSD 2014 member
member(Bm111)	LCOM_501	0.20	OK	0.32	uf6.21	API WSD 2014 member
member(Bm102)	LCOM_501	1.00	OK	0.26	uf6.22	API WSD 2014 member
member(Bm106)	LCOM_501	0.00	OK	0.19	uf6.21	API WSD 2014 member
member(Bm100)	LCOM_501	1.00	OK	0.19	uf6.22	API WSD 2014 member

➢ And displayed:



➢ To learn more about code checking and redesigning members and joints, go to another tutorial.

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