

SESAM TUTORIAL

GeniE

Piled Jacket Analysis

Valid from program version 8.2



Sesam Tutorial

GeniE – Piled Jacket Analysis

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

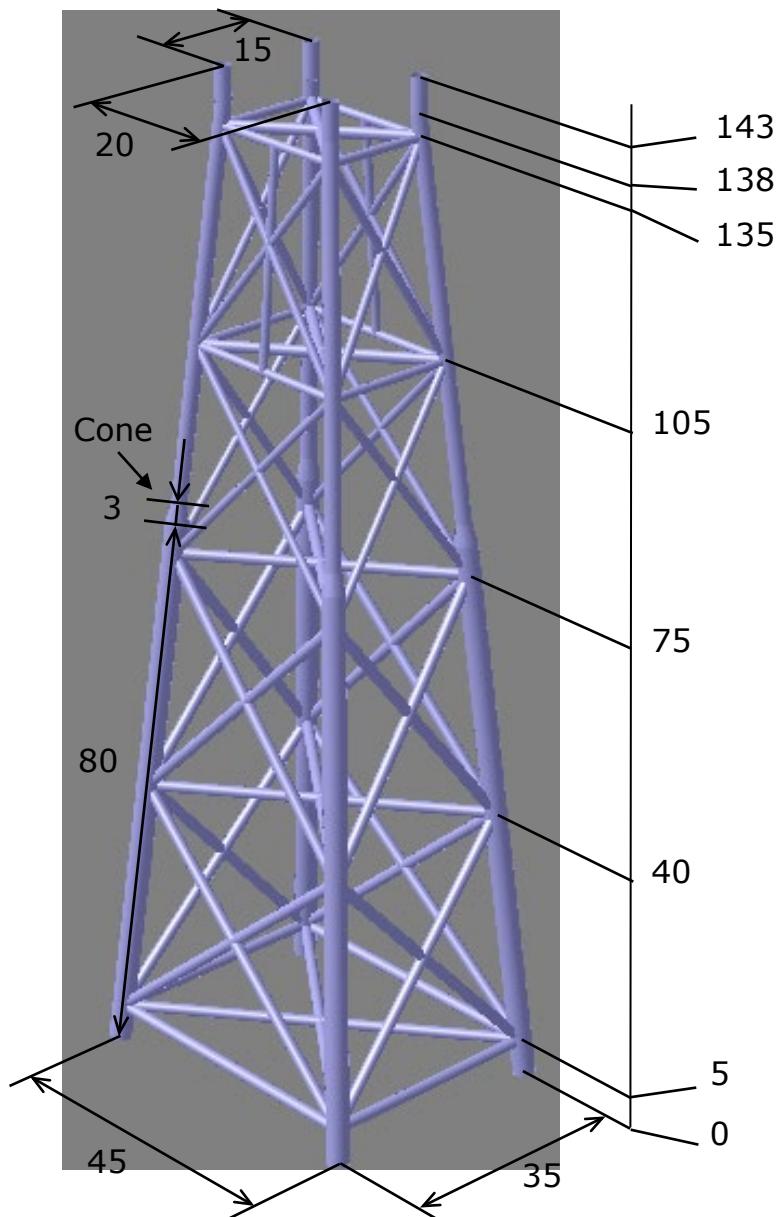
This publication or parts thereof may not be reproduced or transmitted in any form or by any means, including copying or recording, without the prior written consent of DNV AS.

TABLE OF CONTENTS

1. Introduction	Page 4
2. New Workspace, Create Sections, Thicknesses, Materials	Page 5
3. Create Guide Planes	Page 6
4. Create Legs	Page 7
5. Create Bracings	Page 8
6. Change Pipe Section for Upper Part of Legs	Page 13
7. Add Vertical Stubs at Top of Legs	Page 17
8. Create Conductor Supports and Conductors	Page 18
9. Create Simplified Topsides	Page 22
10. Create Sets	Page 24
11. Mesh Property for Topsides Plates	Page 25
12. Create Piles	Page 26
13. Create Soil	Page 28
14. Create Location	Page 33
15. Create Current and Waves	Page 35
16. Create Wave Load Condition	Page 36
17. Create and Assign Hydro Properties	Page 37
18. Create Loads	Page 40
19. Create Wave Load Analysis Activity	Page 42
20. Create Load Combinations	Page 45
21. Run Wave Load Analysis Activity	Page 48
22. Present Results	Page 49

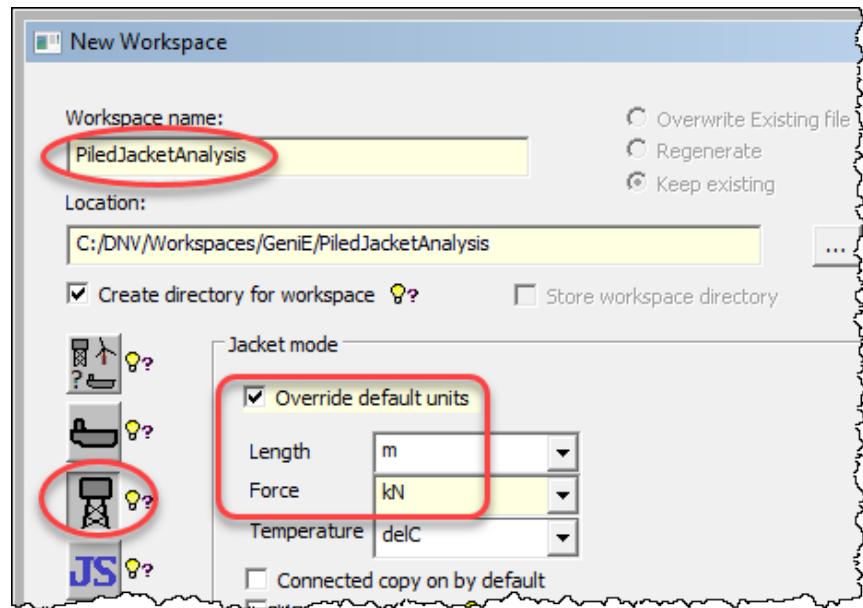
1 INTRODUCTION

- In this tutorial a four-legged jacket with a simplified topside structure is modelled. The jacket is supported by four piles driven through the legs. A static in-place (ULS) analysis is performed with wave loads in addition to self weight.
- The tutorial contains the following steps:
 - Create jacket model with conductors and topside.
 - Model soil and piles.
 - Create a location (hydrodynamic environment) with wave loads.
 - Run a static analysis and do a code checking of the results.
- A GeniE input file for performing all tasks of this tutorial is provided.
- To complete this tutorial you need license to:
 - GeniE code checking
 - Wajac (wave load calculation)
 - Splice (pile-soil analysis)
 - Sestra (structural analysis)
- The jacket with its main dimensions is displayed to the right.



2 NEW WORKSPACE, CREATE SECTIONS, THICKNESSES, MATERIALS

- 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 units m and kN and click *OK*.
- If the workspace name exists select *Overwrite Existing file* or give another name.



- Create the pipe sections shown in the table. The data are given in unit m.
- All sections with *Shear Factors 1* (default) and *Fabrication method Unknown*.

Name	Diameter	Thickness
Pipe06	0.6	0.01
Pipe12	1.2	0.03
Pipe16	1.6	0.03
Pipe21	2.1	0.08
Pipe22	2.2	0.08
Pipe32	3.2	0.09

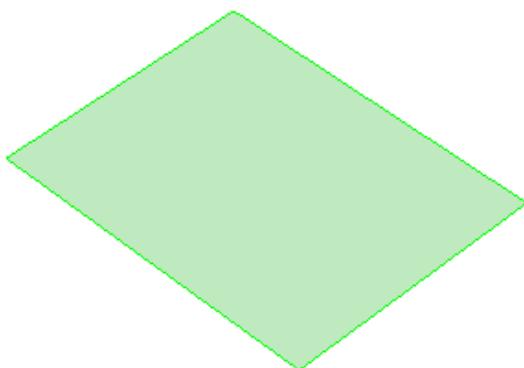
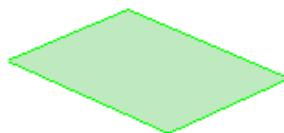
- Create the plate thicknesses shown in the table. The data are given in unit m. The purpose of the very large thicknesses is to compensate for not modelling stiffeners for the plates as will be seen later.
- Create the two linear isotropic materials shown below. The data are given in units compatible with m and kN. PlateMaterial is a special material with very low density to compensate for the very high plate thicknesses. PlateMaterial will only be used for plates.

Name	Thickness
Th2	0.2
Th4	0.4
Th6	0.6

Name	Yield	Density	Young	Poisson	Thermal	Damping
Steel	3.56E5	7.85	2.1E8	0.3	0	0
PlateMaterial	3.56E5	0.785	2.1E8	0.3	0	0

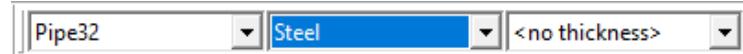
3 CREATE GUIDE PLANES

- Create a guide plane at elevation 138 m which is at top of the legs. The plane spans from -10 m to 10 m in X and from -7.5 m to 7.5 m in Y. Let there be only one spacing in both directions.
- Create another guide plane at elevation 0 m which is at bottom of the legs. The plane spans from -22.5 m to 22.5 m in X and from -17.5 m to 17.5 m in Y. Again, let there be only one spacing in both directions.
- The two guide planes are shown below.



4 CREATE LEGS

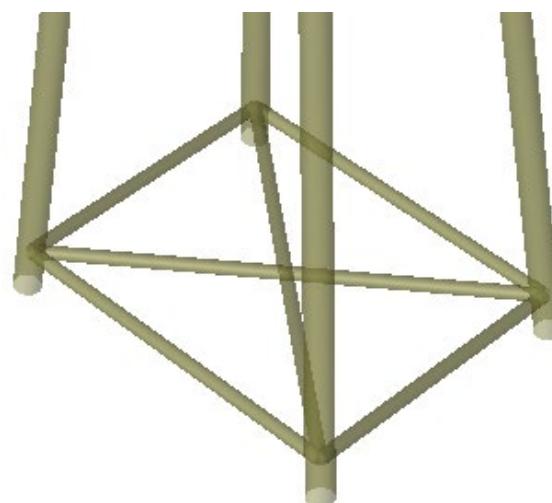
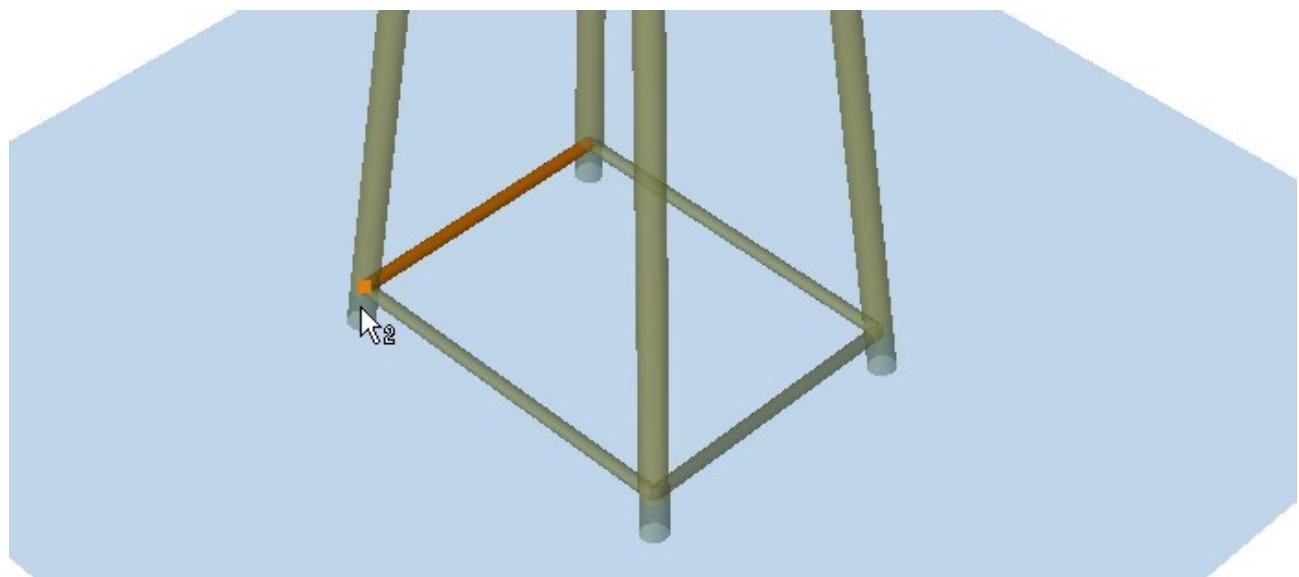
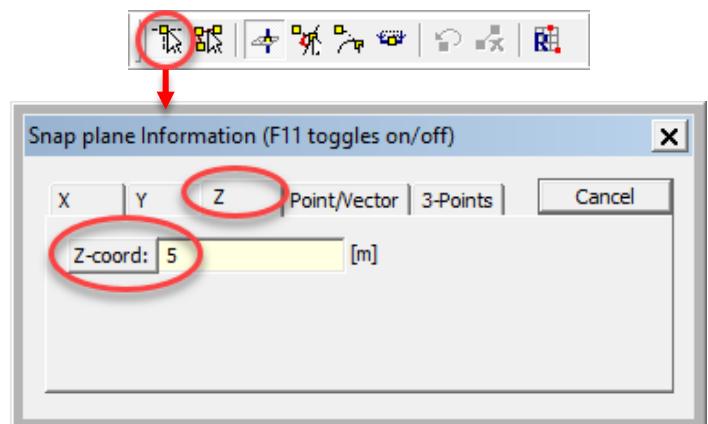
- Set default beam cross section to Pipe32 and default material to Steel and create the four legs.



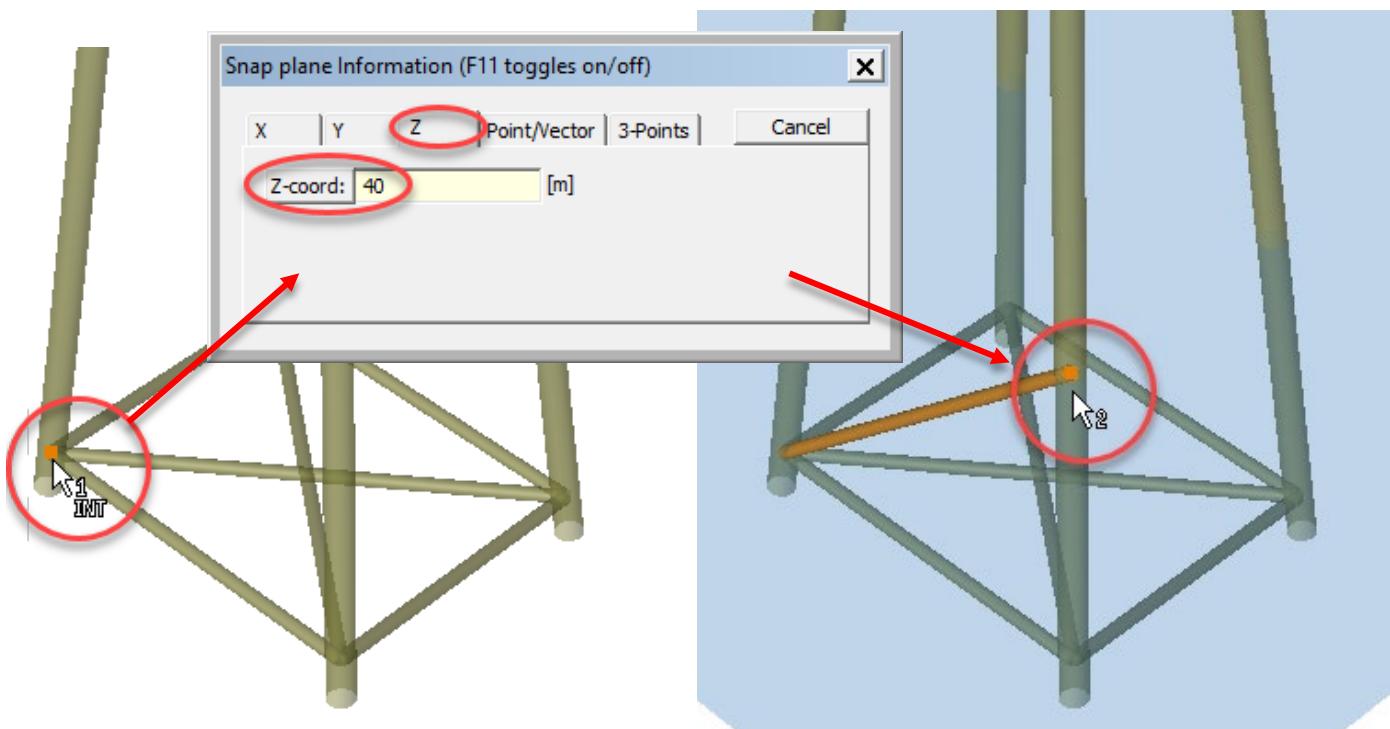
- The guide planes are no longer needed so switch to *Modelling - Transparent* display configuration.

5 CREATE BRACINGS

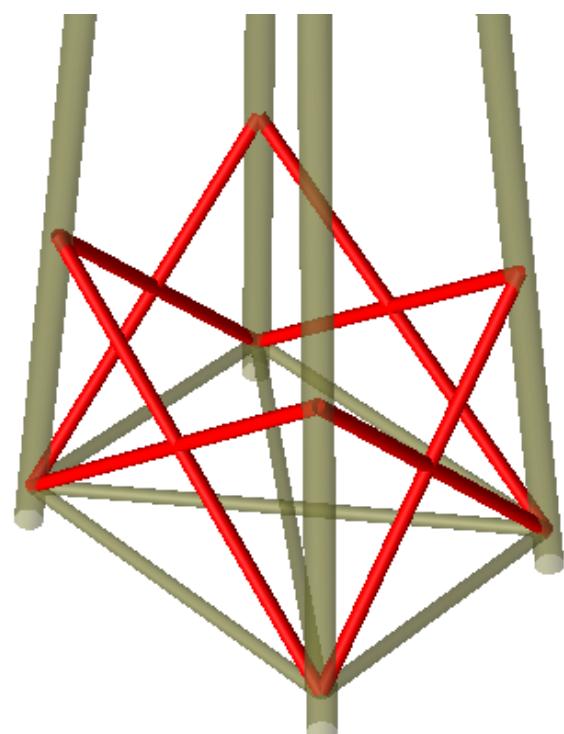
- Bracings are created using horizontal snap planes. When a snap plane is defined and displayed, inserting beams by clicking is restricted to where the snap plane intersects existing beams, i.e. the legs.
- The figure on page 4 defines the elevations of the bracings: 5 m, 40 m, 75 m, 105 m, 135 m, 138 m and 143 m.
- First open a snap plane at elevation 5 m as shown to the right and create the four bracings shown below.
All with pipe section Pipe16.
- Close the snap plane dialog (F11) and add two horizontal diagonal bracings as shown further below.
Also these with pipe section Pipe16.



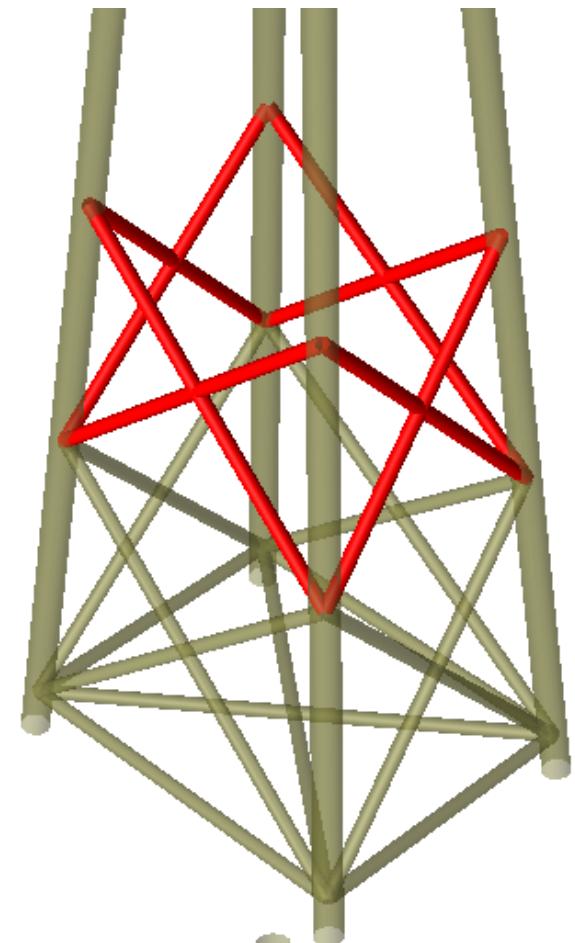
- Create bracings between elevations 5 m and 40 m.
 - Click to position the first end as shown to the left below.
 - Then press F11 to open the snap plane dialog and give Z-coord 40 m.
 - Finally, click to position the second end of the bracing as shown to the right below.



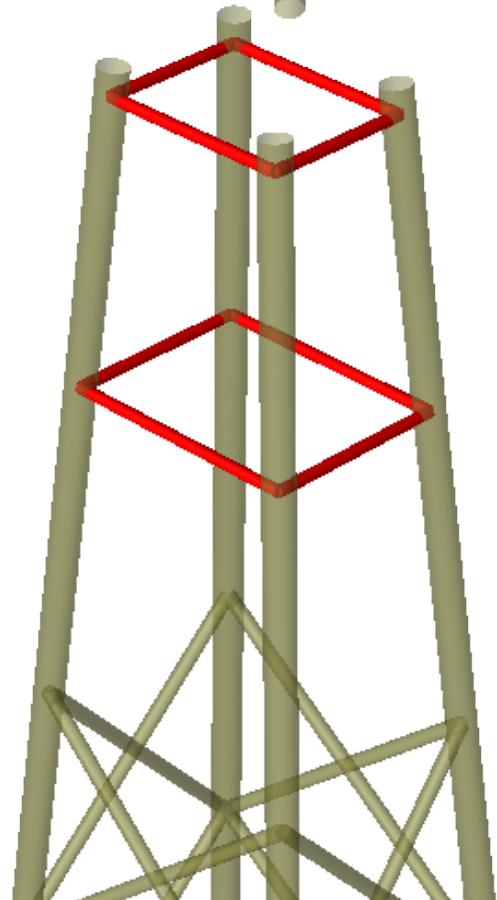
- Use the F11 button to open and close the snap plane dialog, thereby alternating between clicking the lower and upper ends of all bracings between elevations 5 m and 40 m.
- The bracings to create are highlighted to the right.



- Create bracings between elevations 40 m and 75 m. Also these with section Pipe16.
 - Use the same procedure as when creating bracings between elevations 5 m and 40 m.
 - Note that since the beam end positions at elevation 40 m already exist, the *Z-coord* of the snap plane dialog should be set to 70 m.
 - The bracings to create are highlighted to the right.



- Create horizontal bracings at elevations 105 m and 135 m. Use pipe section Pipe12.
 - The bracings to create are highlighted to the right.

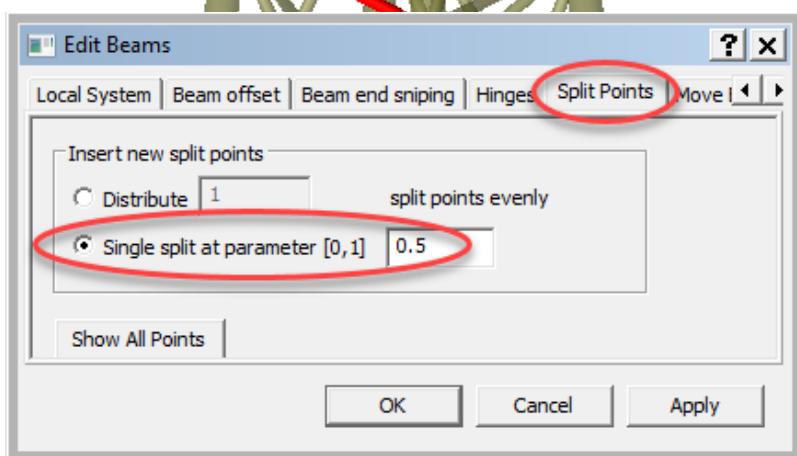
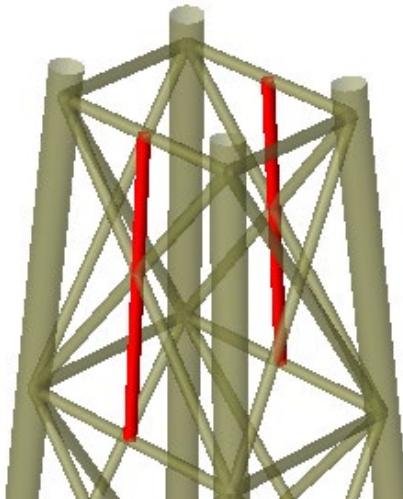
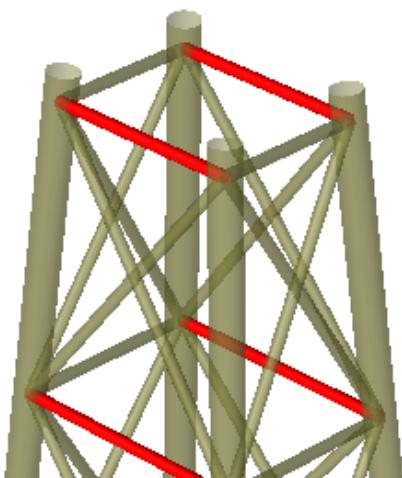
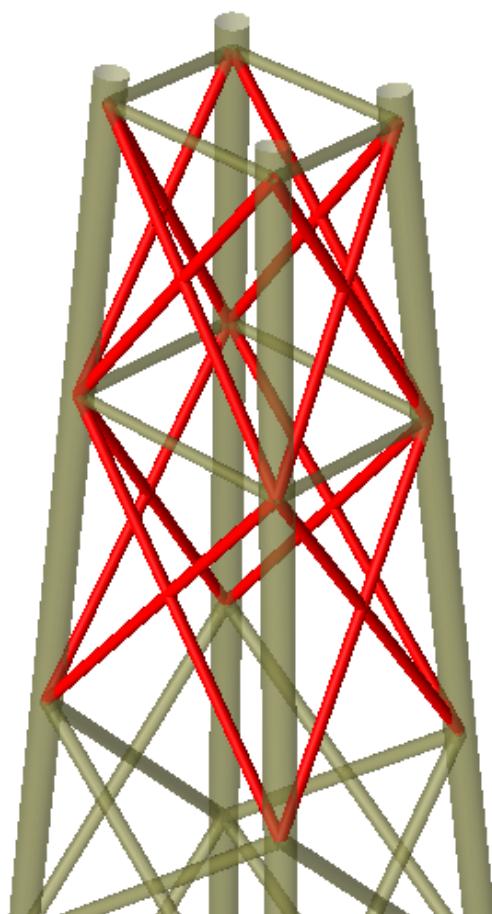


- Create bracings between elevations 75 m and 105 m, and also between 105 m and 135 m. All with pipe section Pipe12.

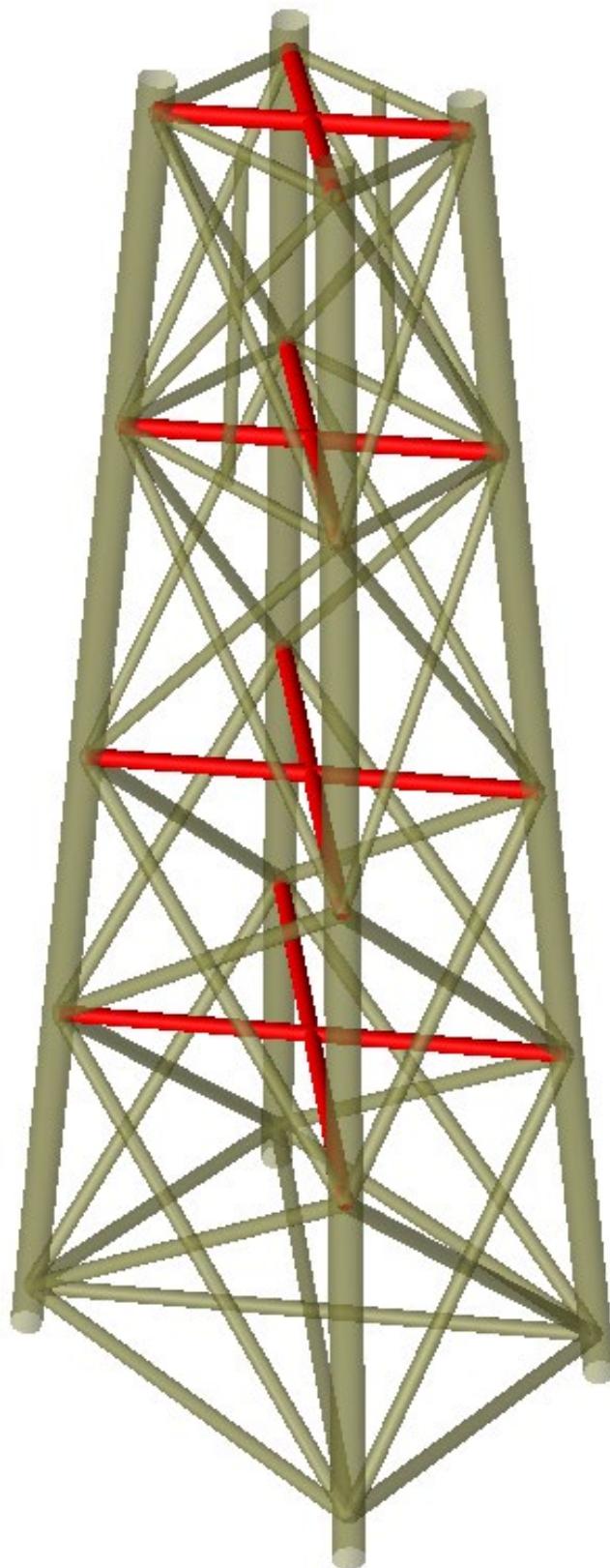
- The bracings to create are highlighted to the right.

- Create the near vertical bracings highlighted in red to the lower right. Their cross sections are Pipe12.

- To insert the beams, first create split points for the four highlighted horizontal bracings shown below. (Creating split points does not mean that the beams are divided.)
- Click the split points to create the two bracings.

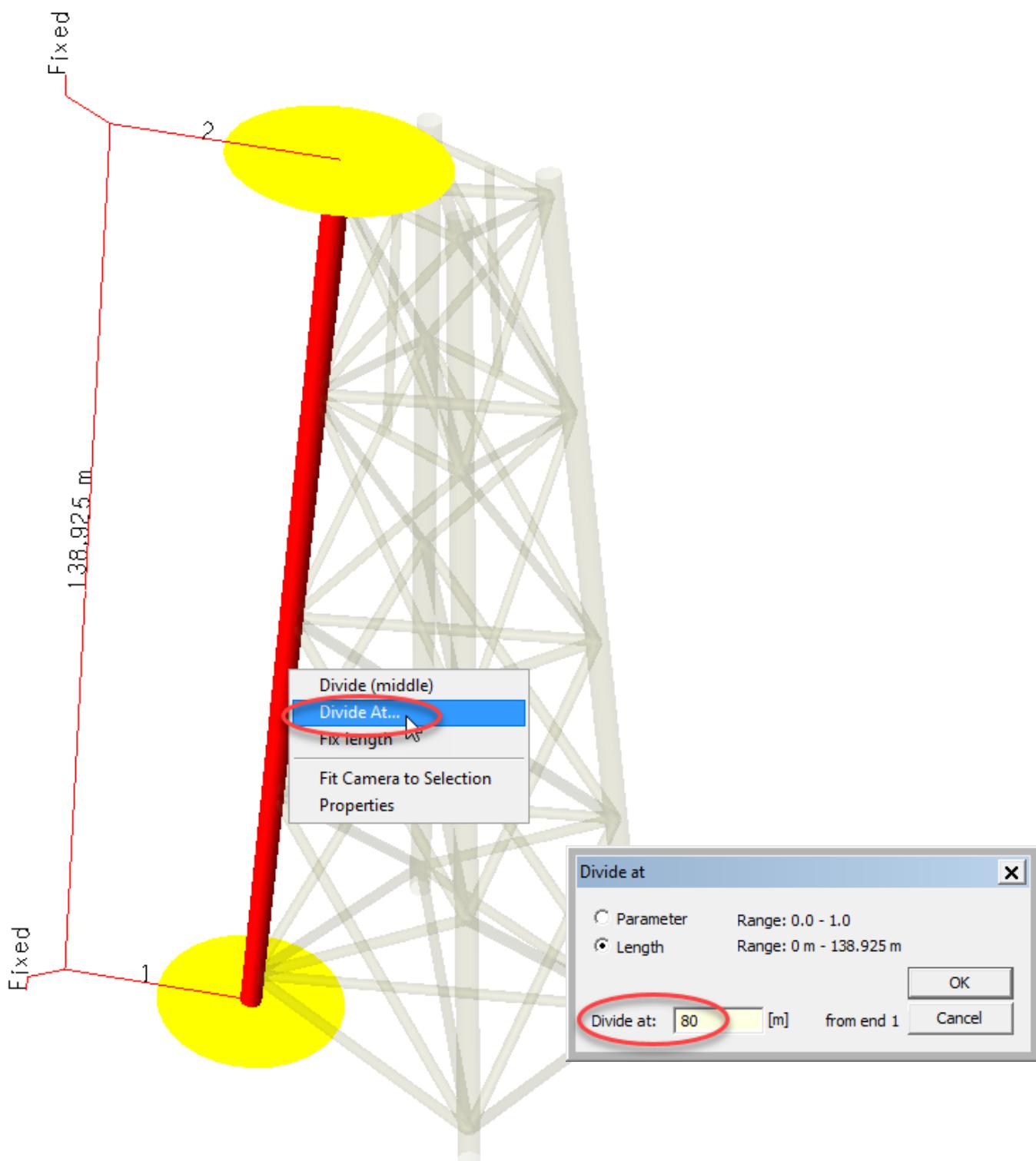


- Create horizontal diagonal bracings at elevations 40 m, 75 m, 105 m and 135 m. All with pipe section Pipe16.

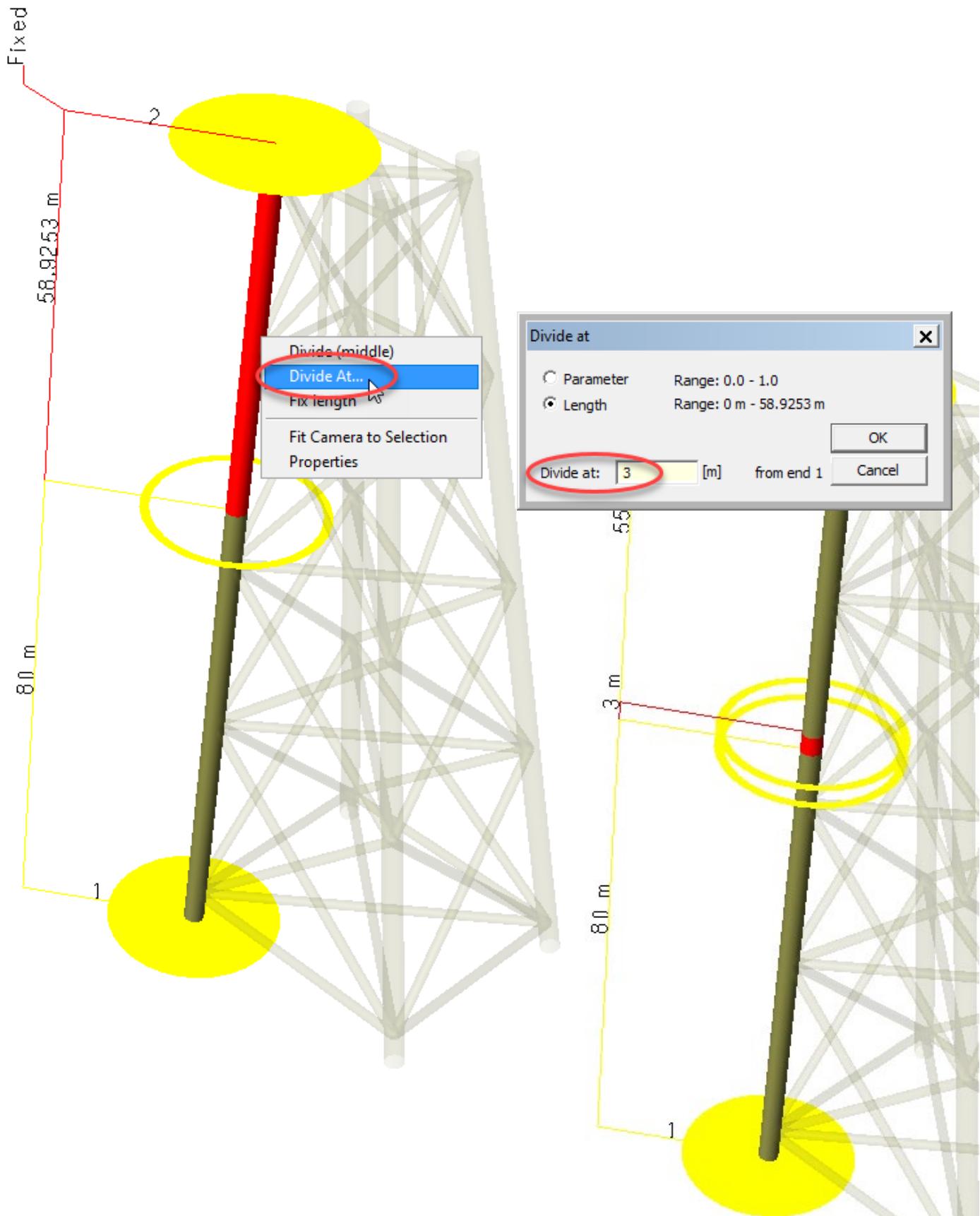


6 CHANGE PIPE SECTION FOR UPPER PART OF LEGS

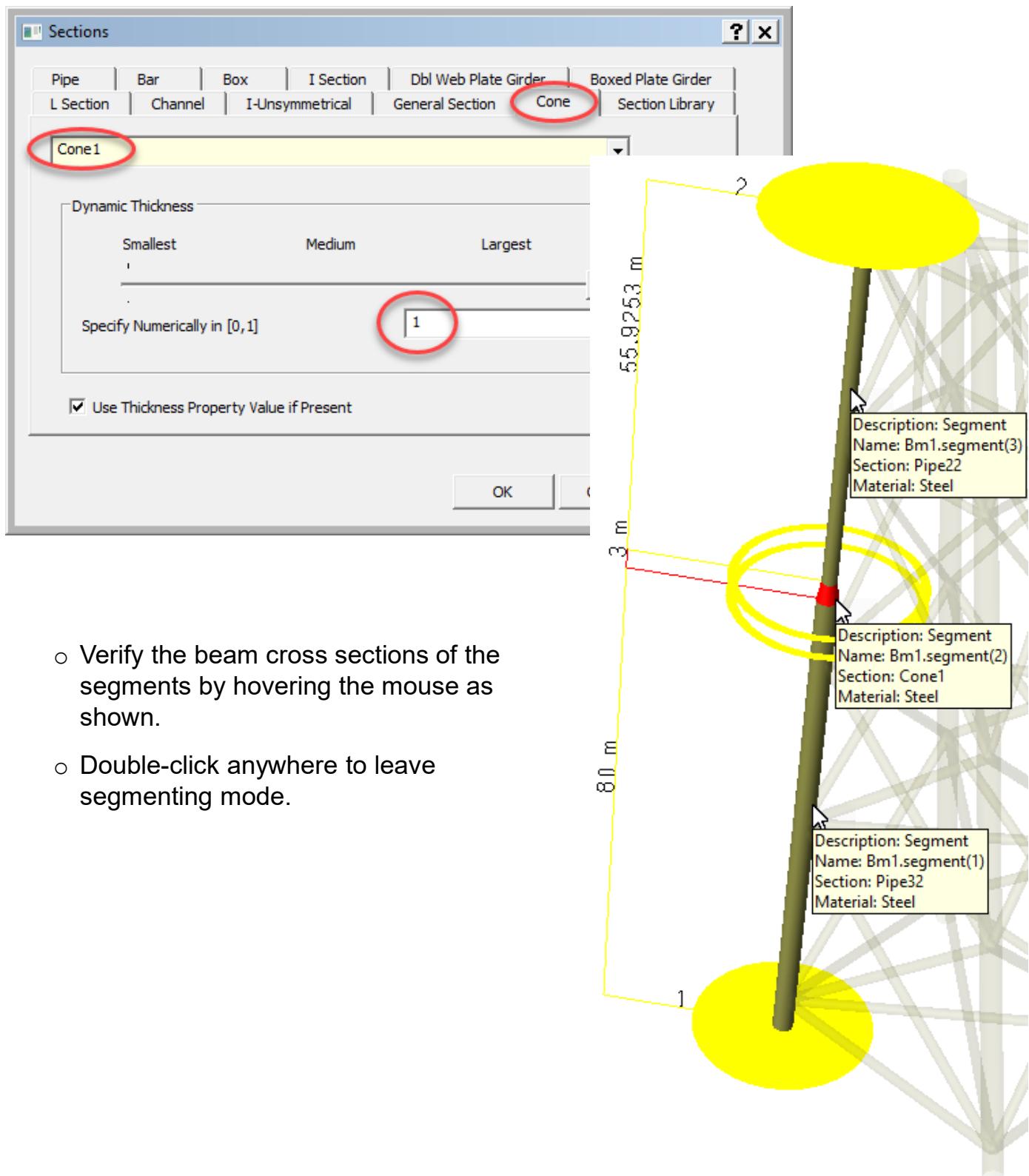
- The four legs have been created with pipe section Pipe32. However, their upper parts shall have pipe section Pipe22. And in between the two pipe sections there shall be a cone.
- Enter segmenting mode by double-clicking a leg.
 - Select and right-click the leg and select *Divide At* to divide it at 80 m measured from the lower end.



- Thereafter right-click the upper part and divide it 3 m measured from the lower end. This will be the cone.

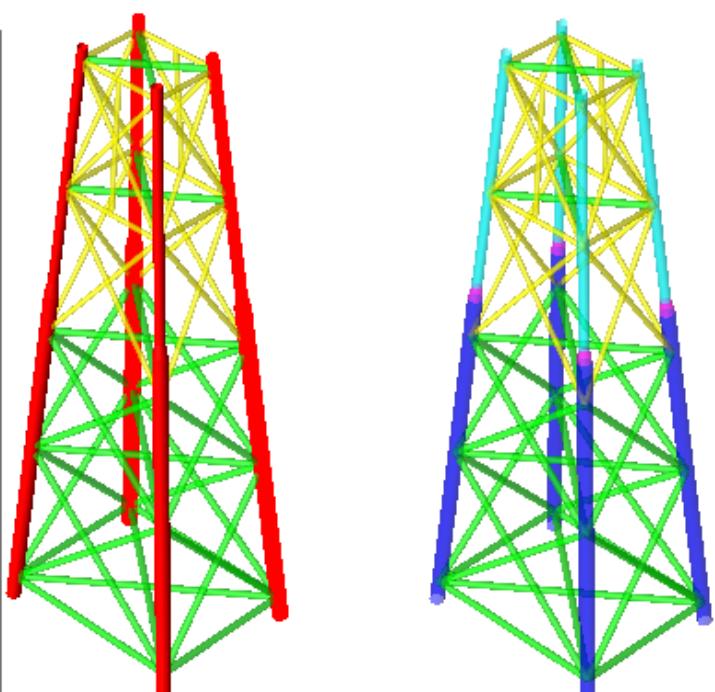
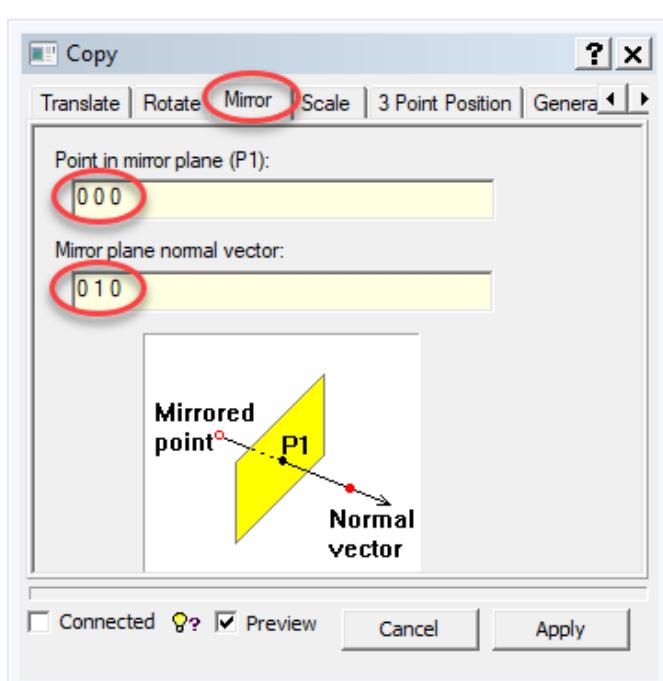
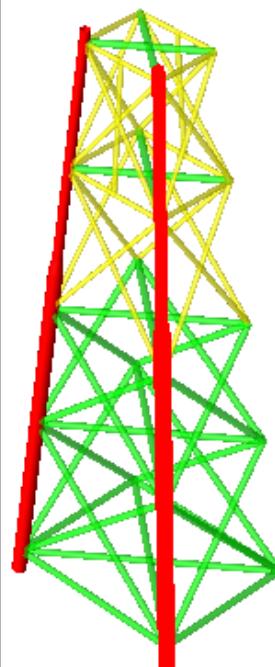
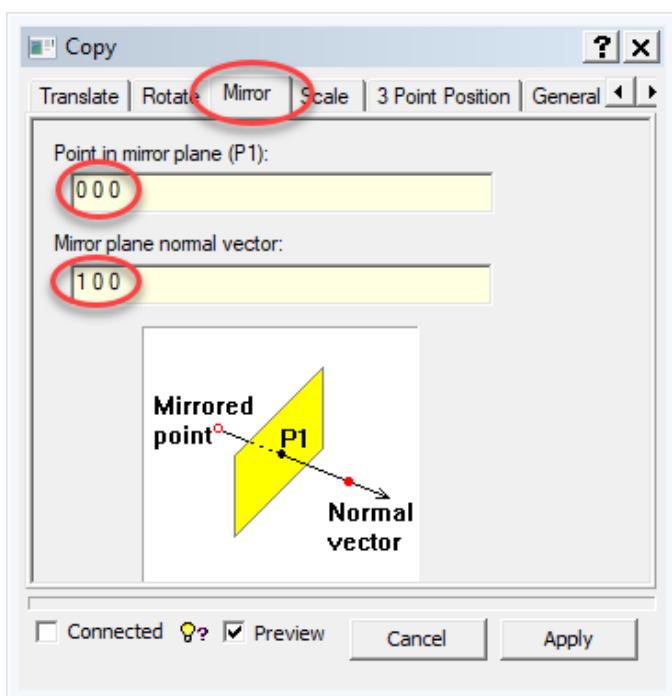


- Assign (apply) pipe section Pipe22 to the upper part.
- The 3 m segment in between shall be a cone. Create such a section as shown below and assign it to the segment. *Dynamic Thickness* set to 1 means that the cone thickness is equal to the larger of the two pipe section thicknesses.



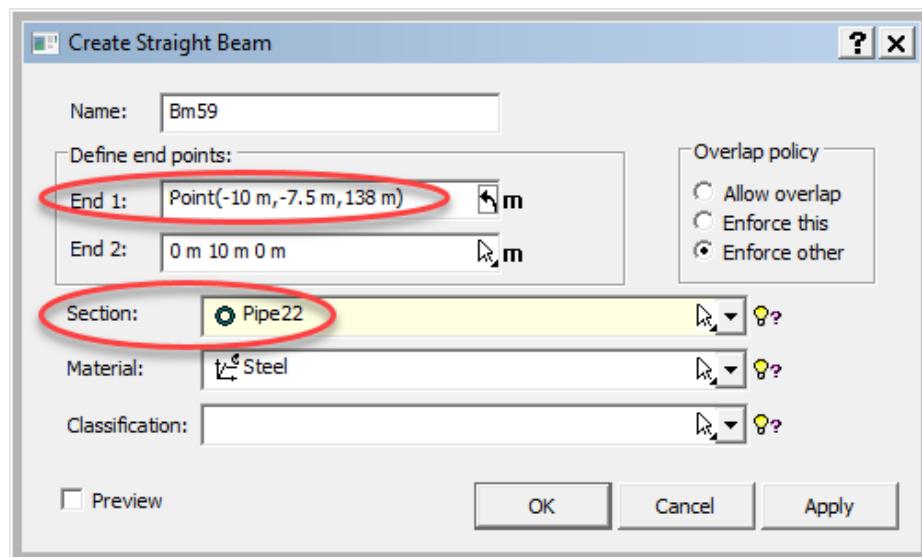
- Verify the beam cross sections of the segments by hovering the mouse as shown.
- Double-click anywhere to leave segmenting mode.

- To change the three other legs either repeat the process above or delete the three other legs and copy the segmented leg by mirroring twice as shown below.
- The fact the X- and Y-axes are 0 at the centre point of the jacket allows copying by mirroring about the origin, first in X-direction and then in Y-direction.
- See that beam cross sections have been colour coded.
- The model with all legs segmented is shown to the lower right.

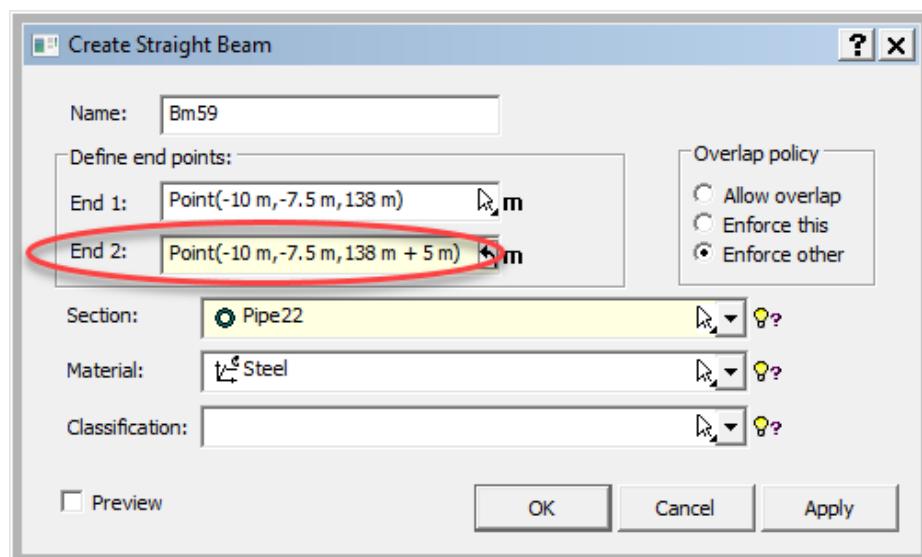


7 ADD VERTICAL STUBS AT TOP OF LEGS

- Add a vertical stub with pipe section Pipe22 at top of a leg. Use *Structure | Beams and Piles | Straight Beam Dialog* to open the *Create Straight Beam* dialog and click the upper end of a leg to position *End 1* of the stub.

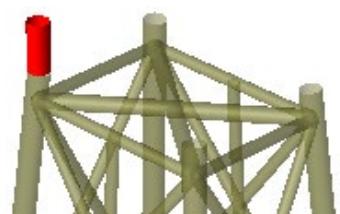
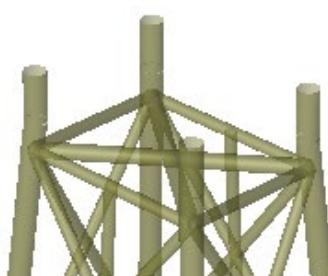


- Copy-paste the *End 1* coordinate into the field for *End 2* and add 5 m to Z.

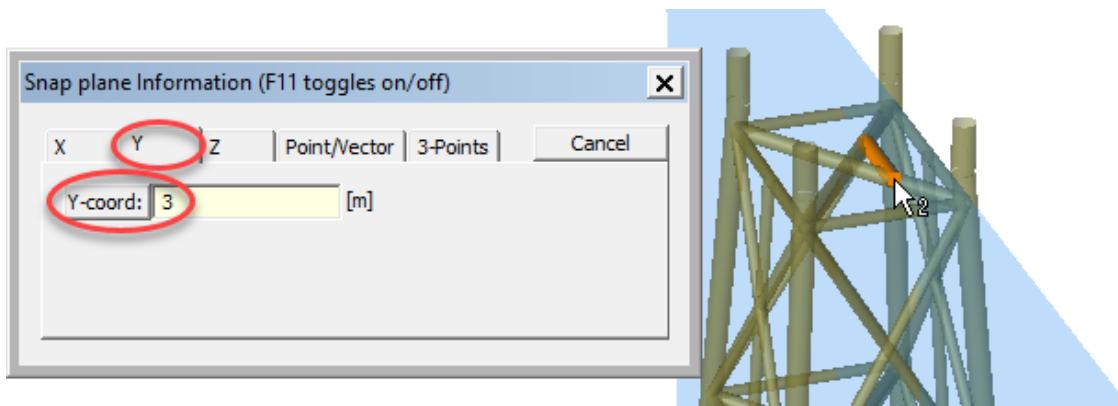


- The stub appears as shown highlighted in red to the right.

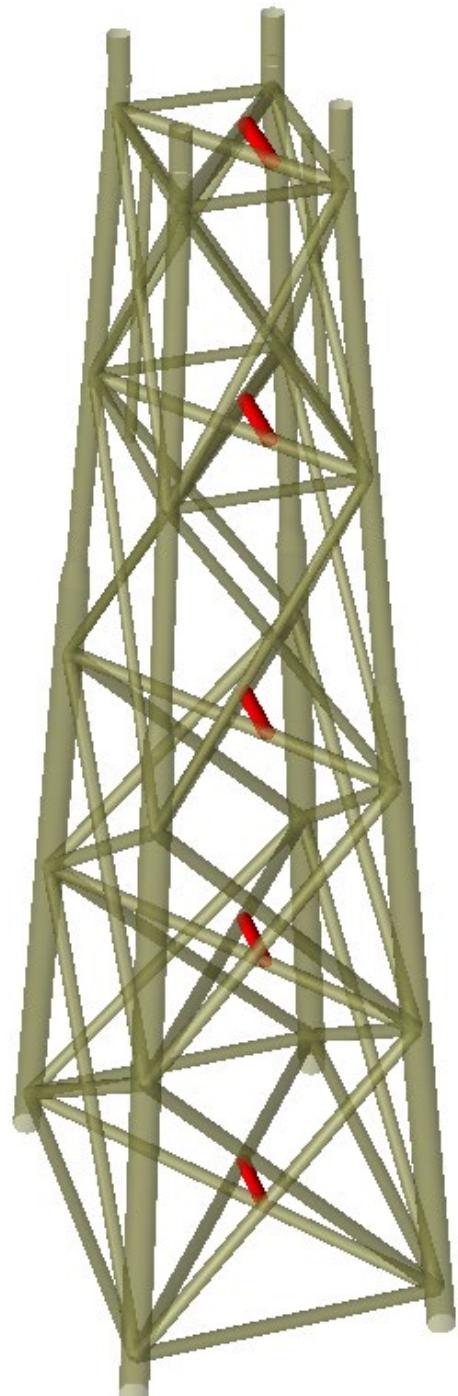
- Copy the stub to the top of the three other legs.



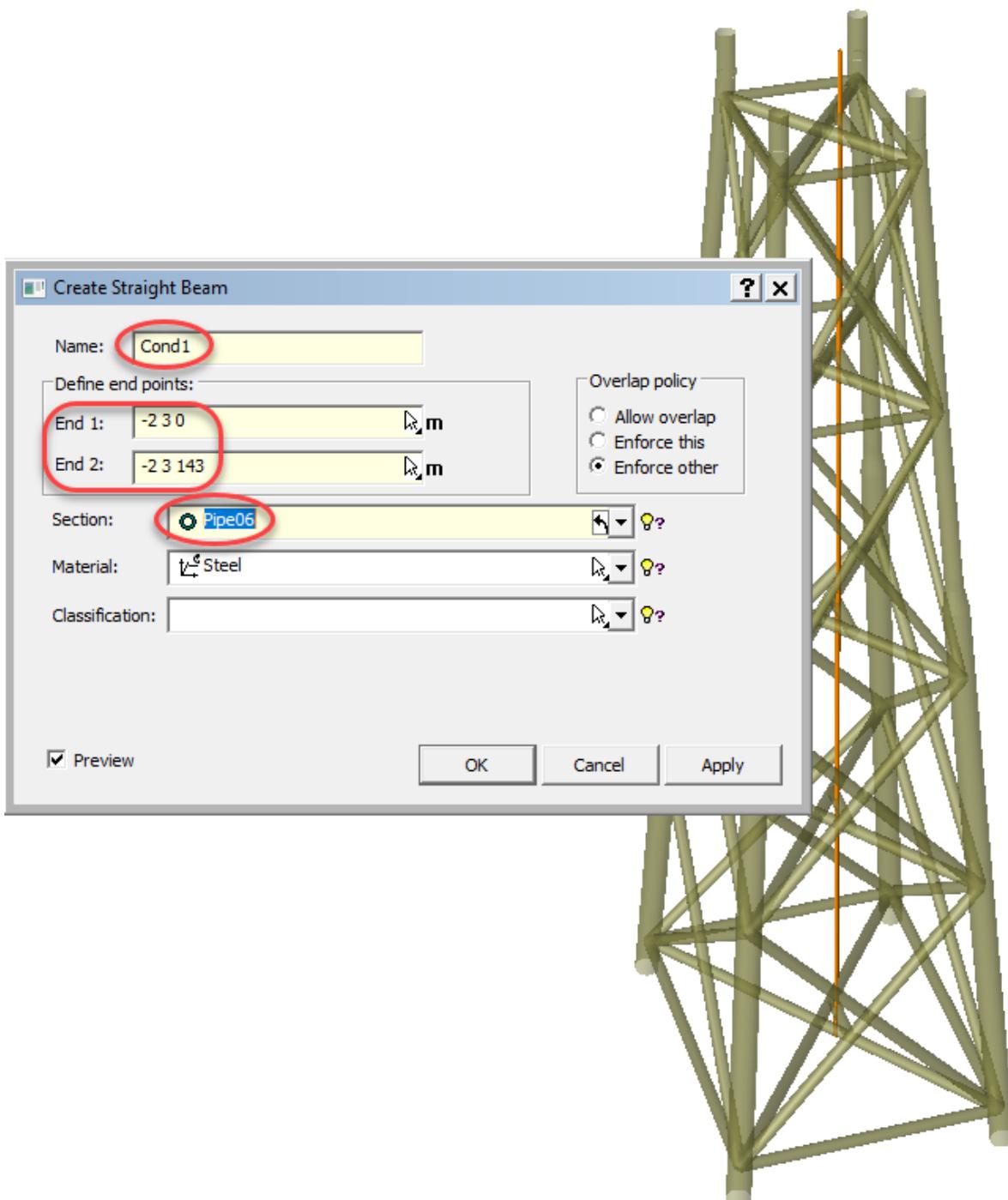
8 CREATE CONDUCTOR SUPPORTS AND CONDUCTORS



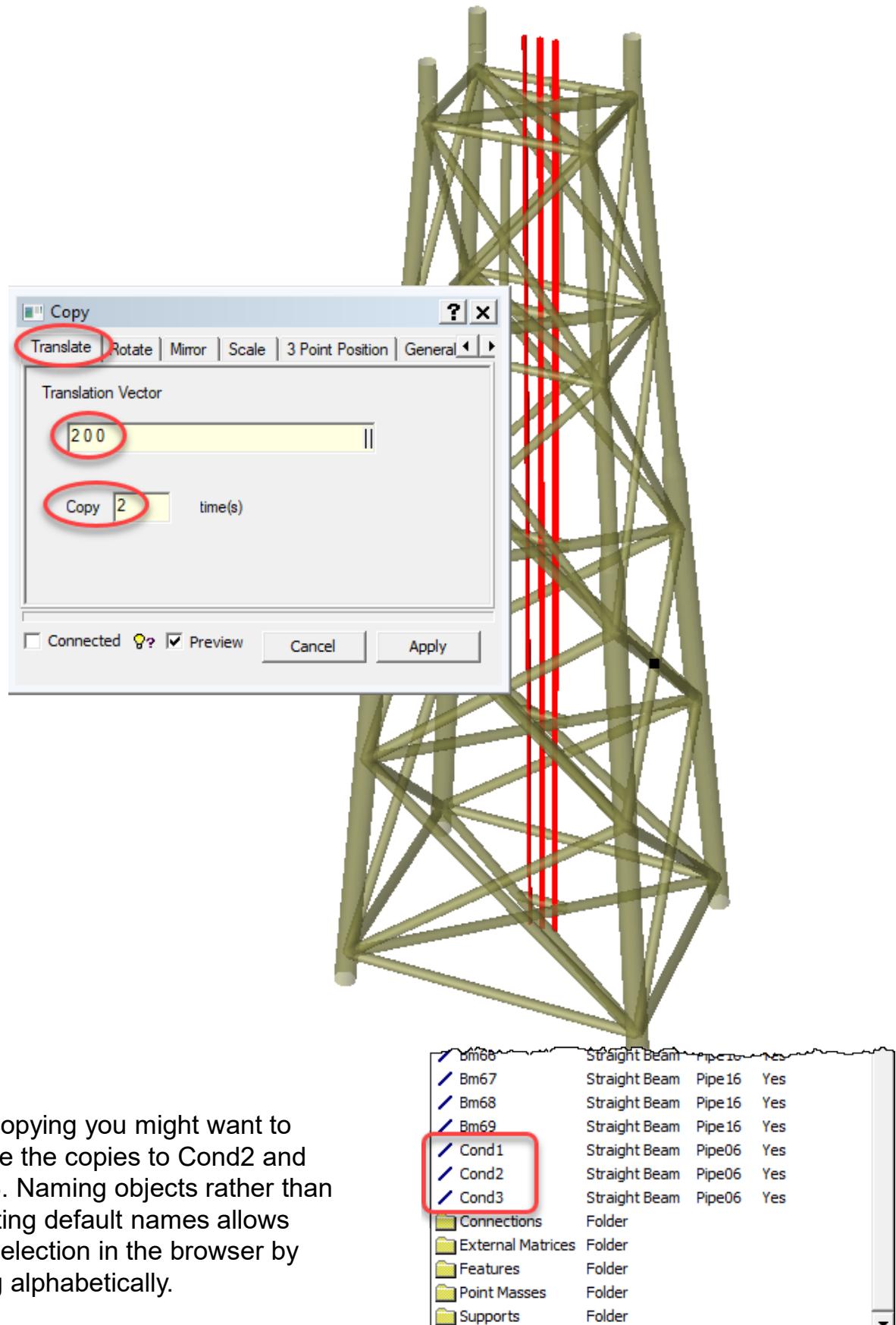
- Three vertical conductors are supported by conductor frames at all elevations.
- The conductors are modelled as non-structural beams, i.e. beams contributing with self weight and hydrodynamic loads but no stiffness to the model.
- The conductor frames are idealized as short horizontal beams at each elevation.
- First, create the short horizontal beams using pipe section Pipe16.
 - Use a vertical snap plane perpendicular to the Y-axis at Y = 3 m.
 - Creating the short horizontal beam at elevation 135 m is shown above.
 - All five short horizontal beams are shown highlighted in red to the right.



- Create one of the conductors by entering its end coordinates directly in the *Create Straight Beam* dialog as shown.
- Let its pipe section be Pipe06.
 - You might want to give it the name Cond1.

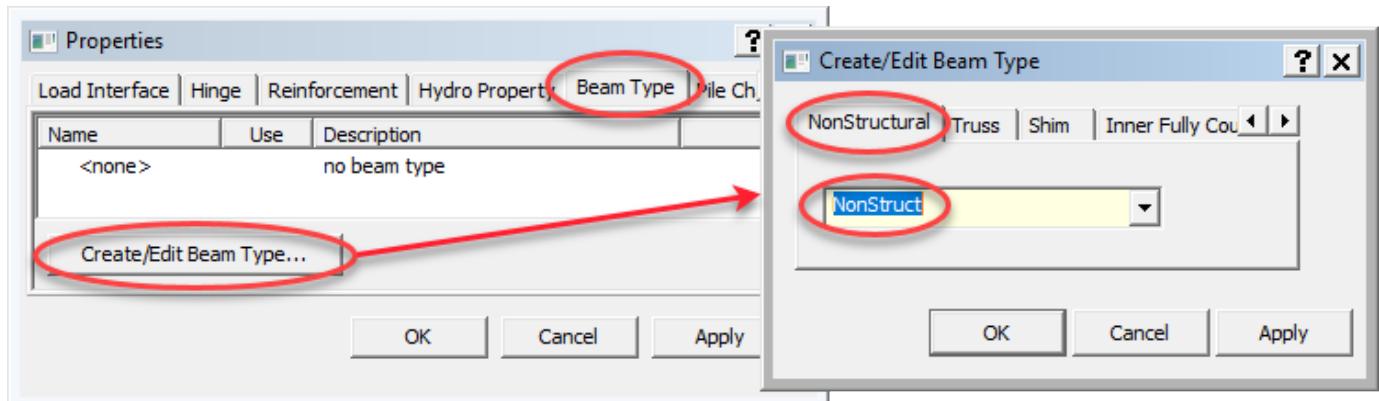


- Copy the conductor twice a distance of 2 m in X-direction.

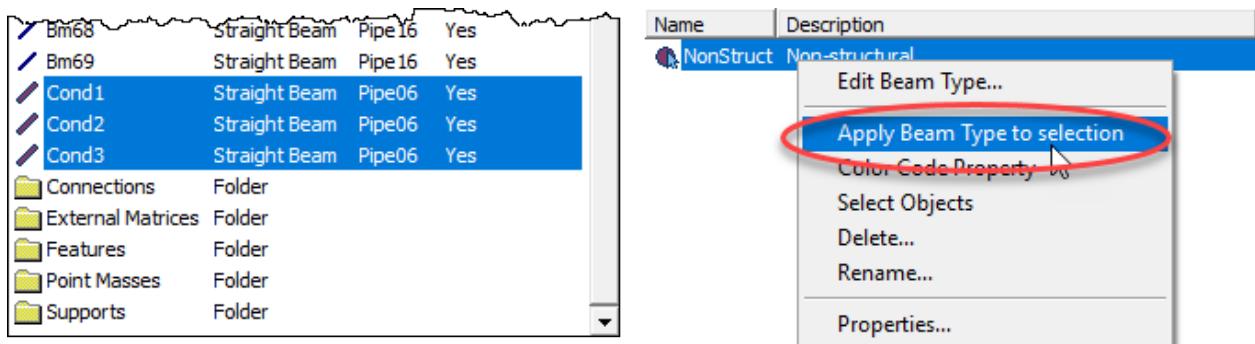


- After copying you might want to rename the copies to Cond2 and Cond3. Naming objects rather than accepting default names allows easy selection in the browser by sorting alphabetically.

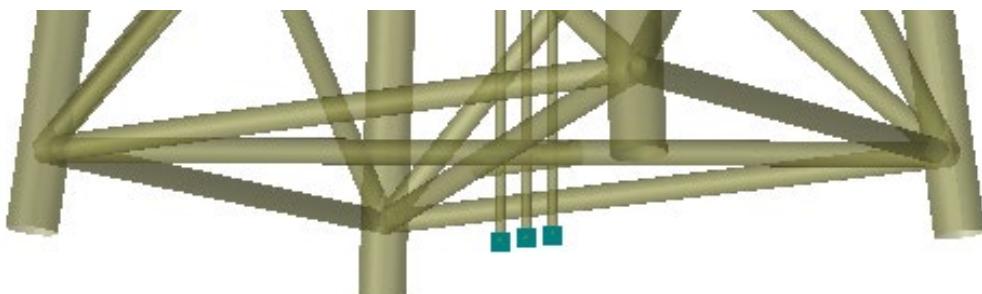
- In the *Properties* dialog go to the *Beam Type* tab and click *Create/Edit Beam Type* to create a *NonStructural* beam type named e.g. *NonStruct*.



- Then select the three conductors and assign the beam type to these.



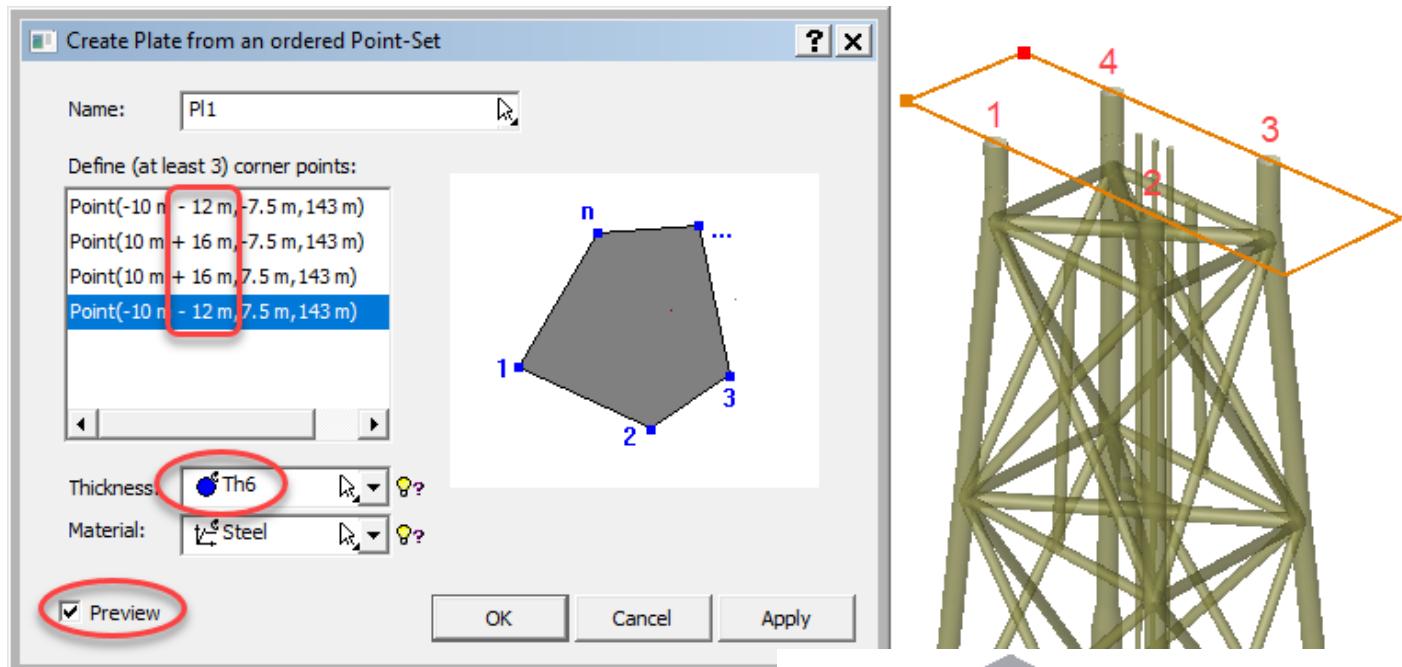
- The lower ends of the conductors must be fixed so that half of the loads on the parts of the conductors below the lowest elevation at 5 m is picked up by the sea floor.



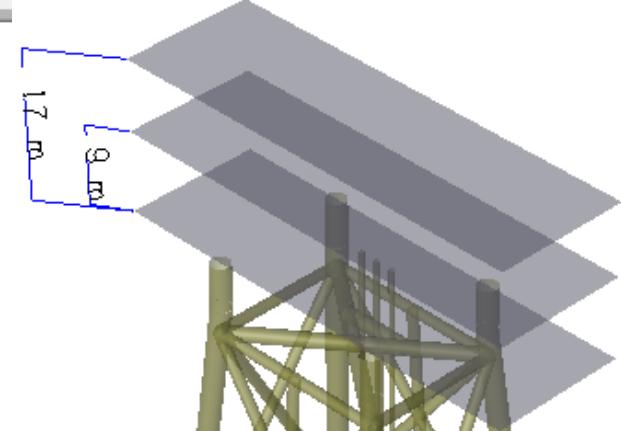
- Before proceeding it may be a good idea to save the workspace (Ctrl+S) and also verify the model by *Structure | Topology | Verify Model* to ensure the consistency of the geometry. Do this from time to time.

9 CREATE SIMPLIFIED TOPSIDE

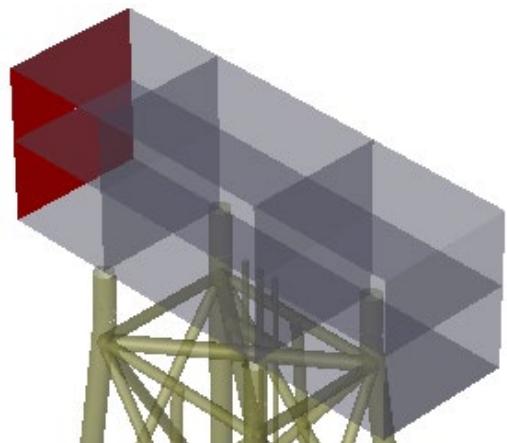
- Use *Structure | Flat Plates | Flat Plate Dialog* to create the lower deck of the topside. First click the top of the stubs as indicated below. Then edit the X values of the four plate corners by subtracting 12 m and adding 16 m as shown. The plate thickness property shall be Th6 and the material property PlateMaterial.



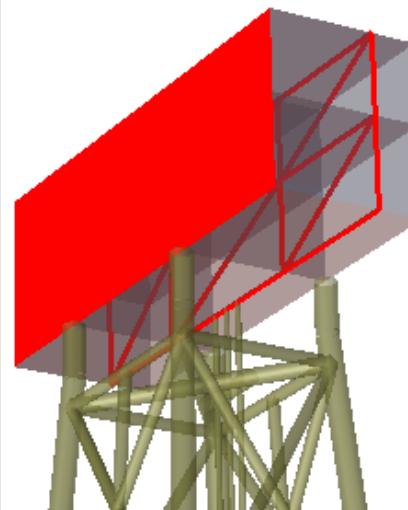
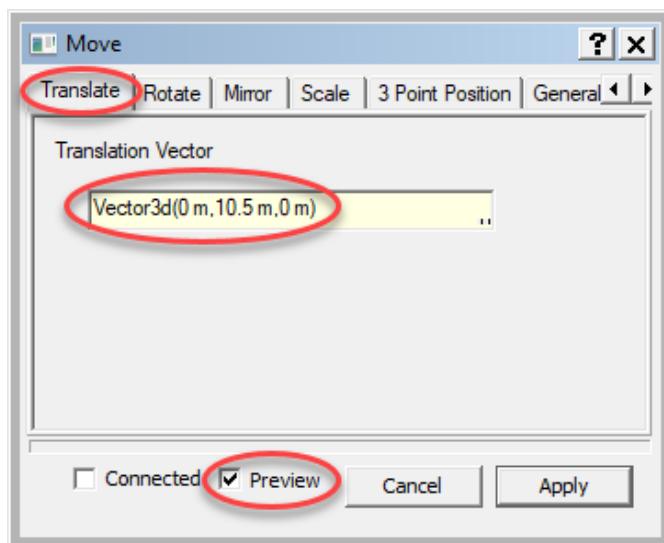
- Copy the lower deck plate 9 m upwards to create the middle deck plate and 17 m upwards to create the upper deck plate.



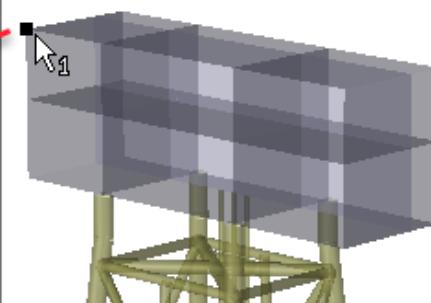
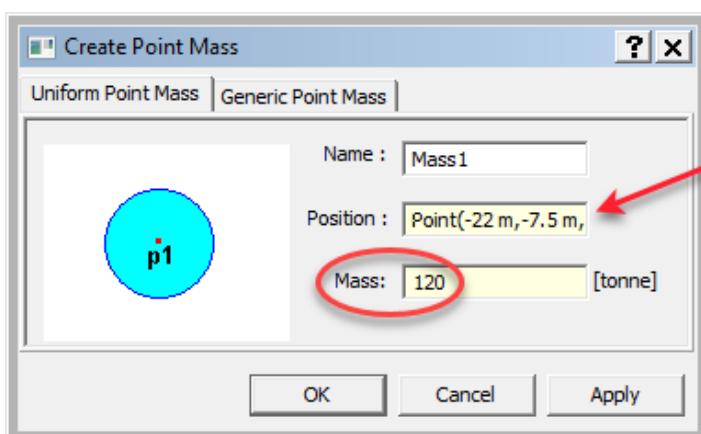
- Create the wall highlighted in red and then copy it to create the other three walls. The two middle walls are positioned at top of the stubs. All walls shall have thickness Th4 and material PlateMaterial.



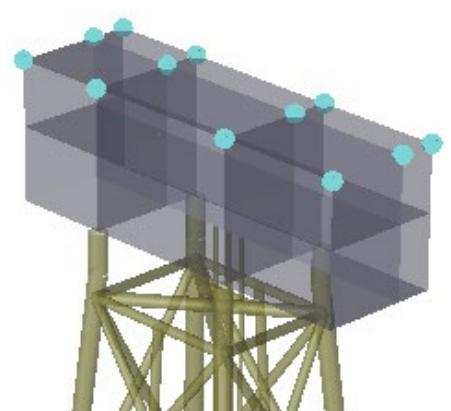
- Create a wall in the XZ-plane with Y-coordinate corresponding to the conductors. The thickness property shall be Th2 and material property PlateMaterial.
- A convenient way of creating the wall is to create it at the edge of the decks and then move it in Y-direction.



- Use *Structure | Point Mass* to add point masses at the shown positions. All of 120 tonnes.

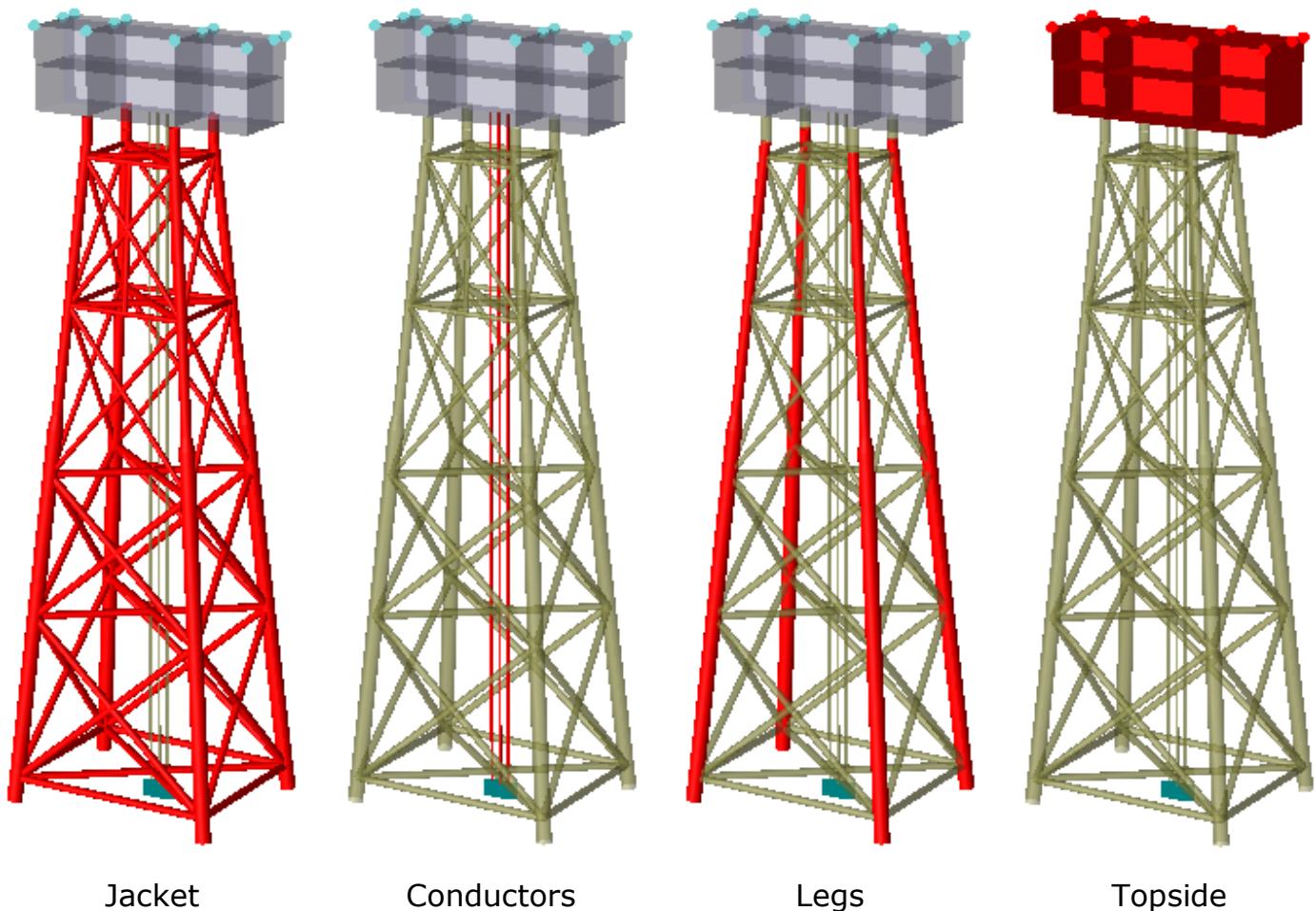


- The 12 point masses are displayed as blue spheres.



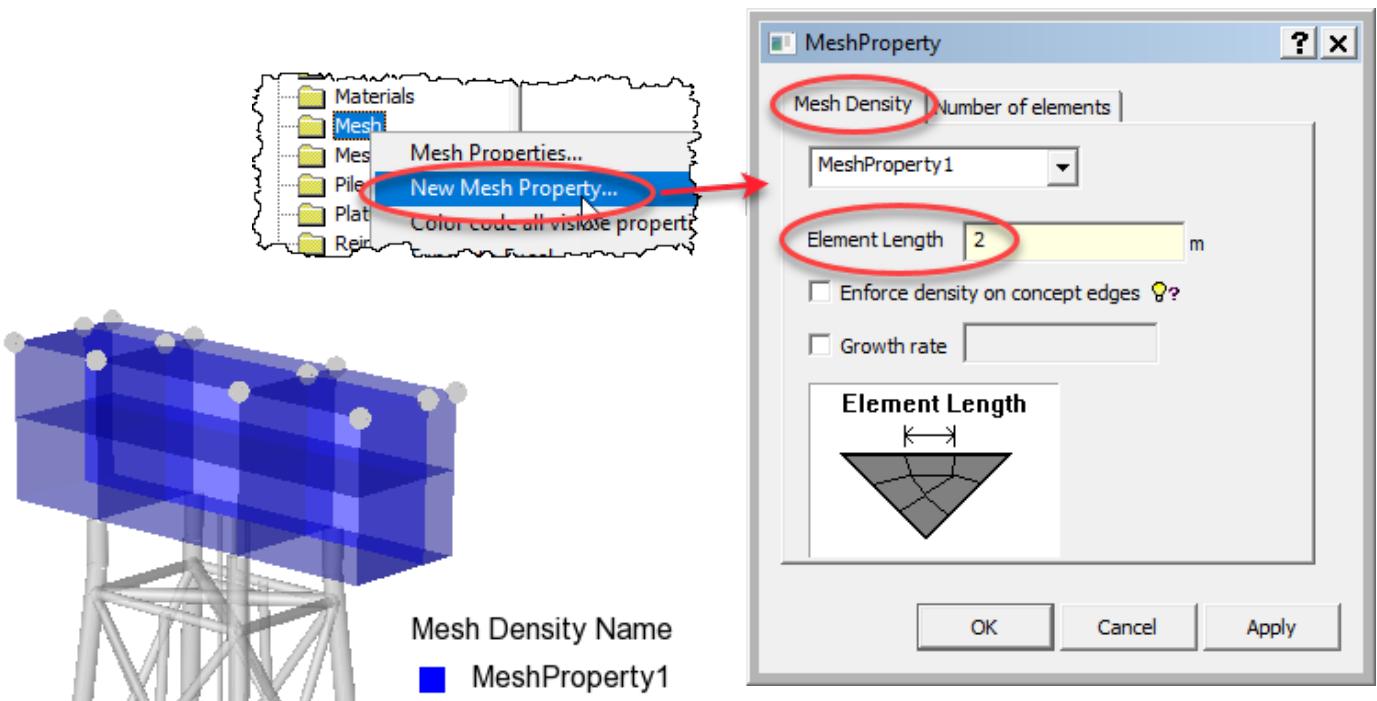
10 CREATE SETS

- Sets are often convenient during modelling and also later in the analysis process.
- Sets are created by making a selection graphically or in the browser, right-clicking, selecting *Named set* and giving a name in the dialog popping up.
- Sets created are stored in the *Utilities | Sets | Regular Sets* folder in the browser.
- Create the following four sets:
 - Jacket
All beams except the conductors
 - Conductors
The three non-structural beams
 - Legs
The four legs
 - Topsides
The plates and mass points

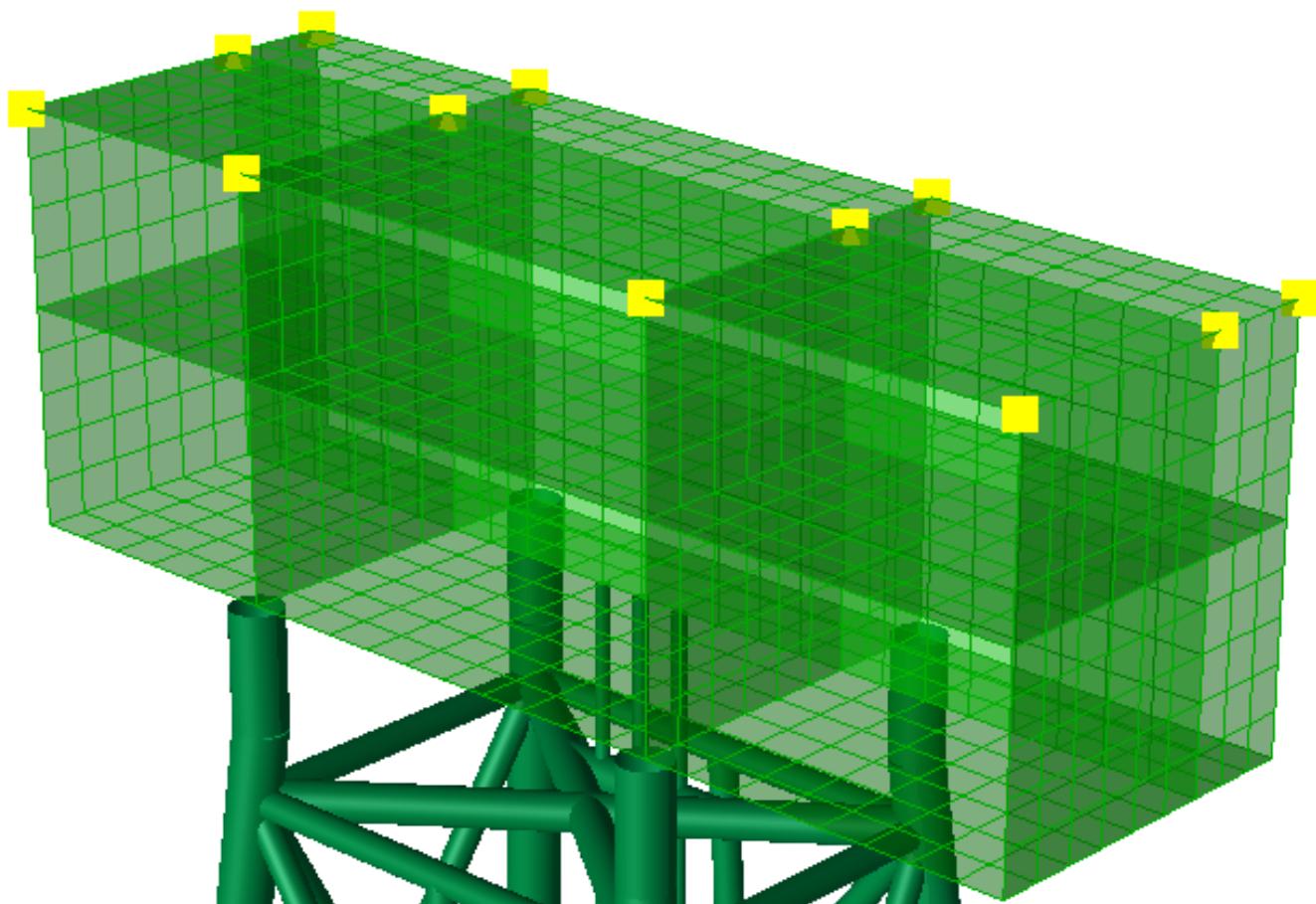


11 MESH PROPERTY FOR TOPSIDE PLATES

- Create a mesh density of 2 m and assign to all plates (not to any beams).

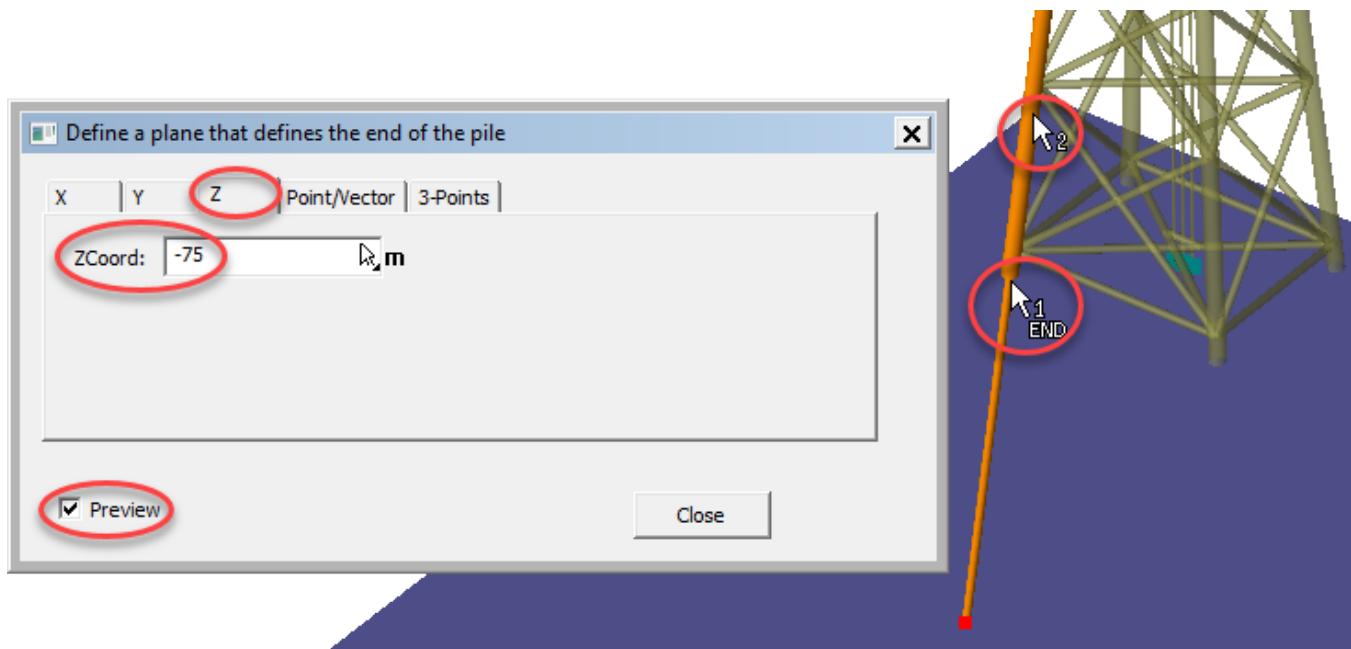


- Use *Mesh & Analysis | Create Mesh* (or Alt+M) to create a mesh merely to verify a suitable mesh for the topside. The mass points appear as yellow squares.



12 CREATE PILES

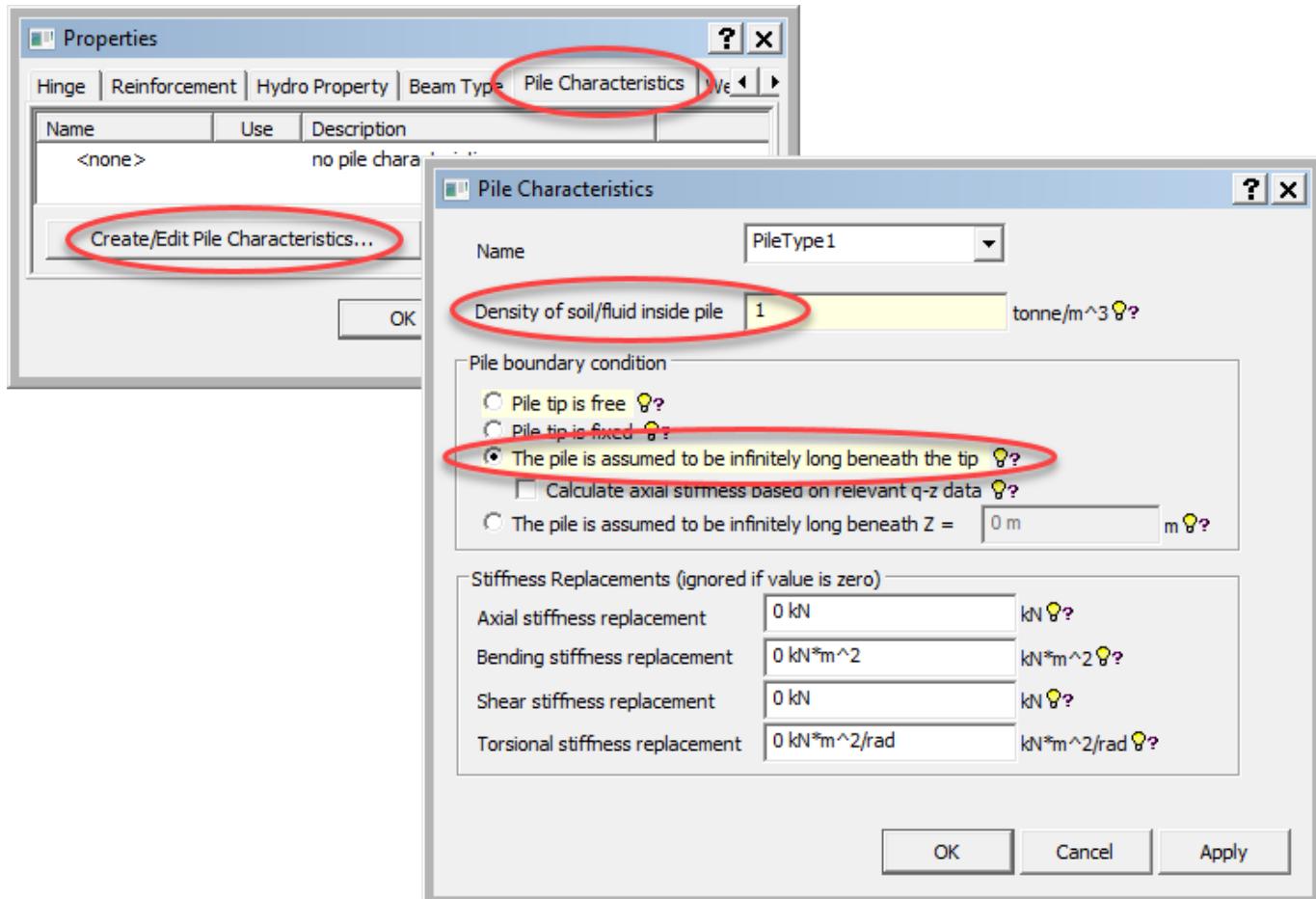
- Piles in the soil are beams with a *Pile Characteristics* property. The piles shall have pipe section Pipe21.
- The part of the piles inside the legs are neglected in this simplified analysis.
- Piles in the soil are created as follows:
 - Use *Structure | Beams and Piles | Pile* and then click the lower end of a leg.
 - A dialog opens for entering the Z-coordinate for the pile end, enter -75 m.
 - Hover the pointer over a leg to align the pile and see a preview.
 - Click the appropriate leg to create the pile.



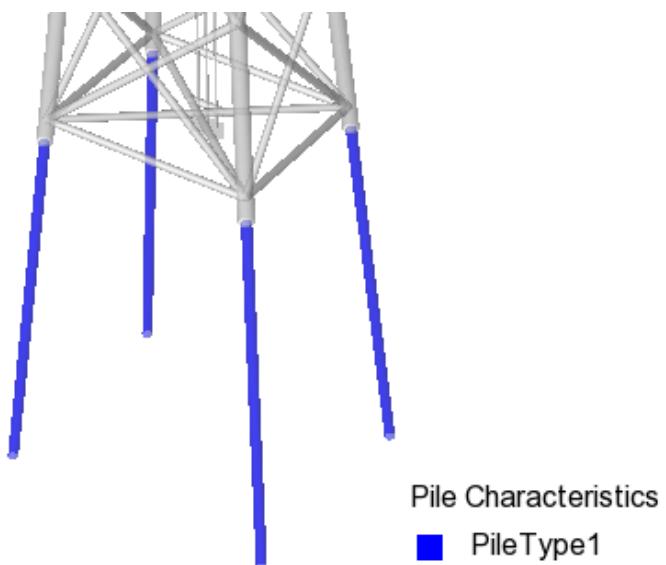
- Create the four piles shown.



- In the *Properties* dialog, go to the *Pile Characteristics* tab and click *Create/Edit Pile Characteristics* to open the *Pile Characteristics* dialog. In this dialog give *Density of soil/fluid inside pile* 1 tonne/m³. Also check *The pile is assumed to be infinitely long beneath the tip*. Read about this pile boundary condition in the Splice user manual.

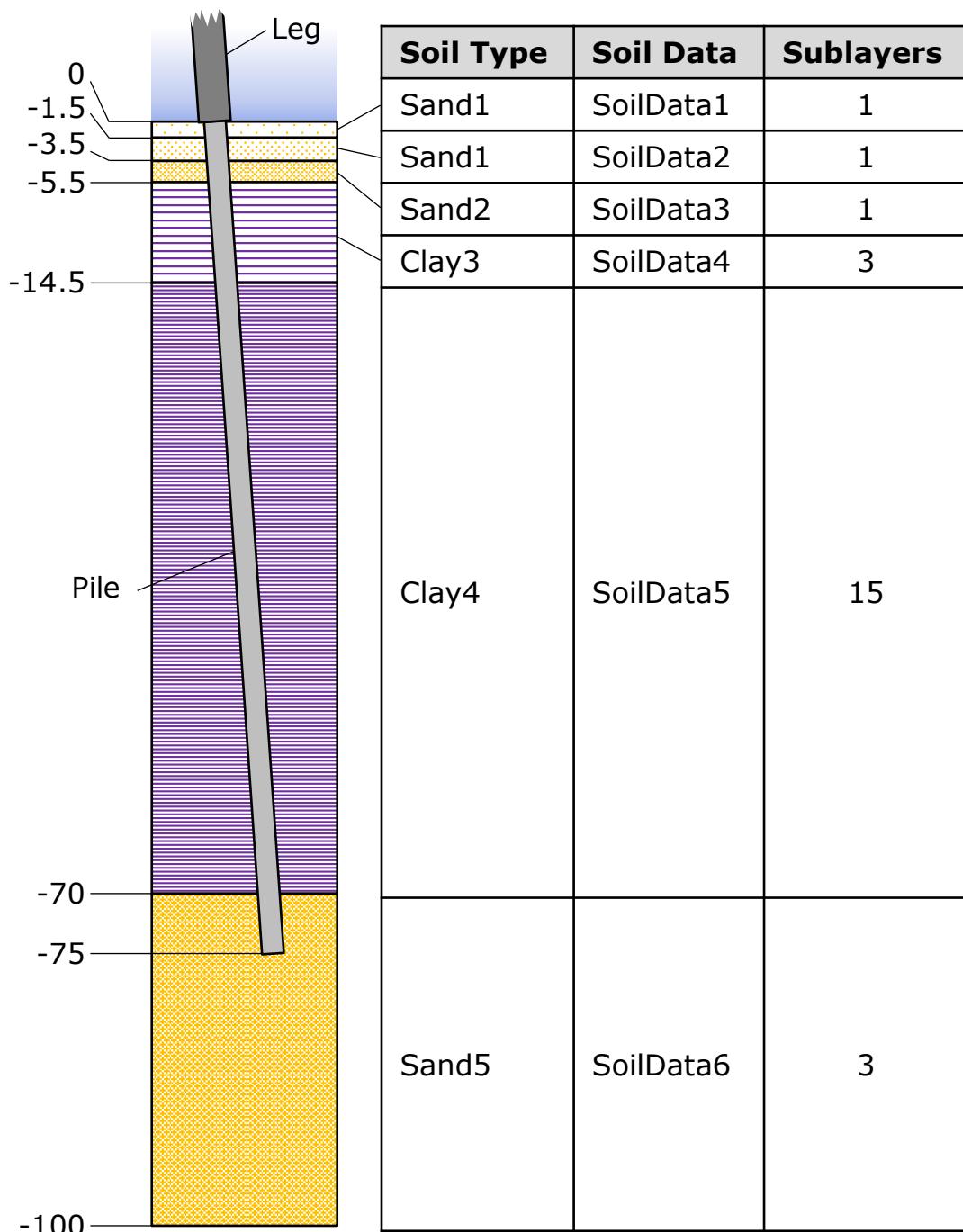


- Assign the property to the four piles.



13 CREATE SOIL

- The soil consists of six layers of sand and clay as shown below.
- Data for the sand and clay soil types are given in two tables next page.
 - Additional soil data are given in a table next page.
 - The thick soil layers are divided into sublayers to refine the analysis.



- Data for the sand and clay soil types are found in the two tables below. All values in units consistent with m and kN.

Soil type	Angle internal friction	Density	Open gap?	Over-consolidation ratio	Residual/peak skin friction ratio
Sand1	40	1.99	No	1	1
Sand2	36	1.99	No	1	1
Sand5	37	2.04	No	1	1

Soil type	Suz1(0)	Suz2(-100)	Density	Strain at half max stress	J-factor	Open gap?	Over-consolidation ratio	Residual/peak skin friction ratio
Clay3	200	100	1.94	0.01	0.5	No	1	1
Clay4	300	130	1.94	0.01	0.5	No	1	1

- Additional soil data are given in a table below. All values in units consistent with m and kN.

Soil data	Initial value of soil shear modulus	Soil Poisson ratio	Peak skin friction				Tip resistance	
			Compression	Tension	Ratio betw. displ. to reach peak skin friction and pile diam.	Peak tip stress	Ratio betw. displ. to reach peak tip stress and pile diam.	
SoilData1	-1	0.5	5	3	0.01	-	-	
SoilData2	-1	0.5	15	11	0.01	-	-	
SoilData3	-1	0.5	45	45	0.01	-	-	
SoilData4	-1	0.5	200	200	0.01	-	-	
SoilData5	-1	0.5	250	250	0.01	-	-	
SoilData6	-1	0.5	120	120	0.01	30000	0.05	

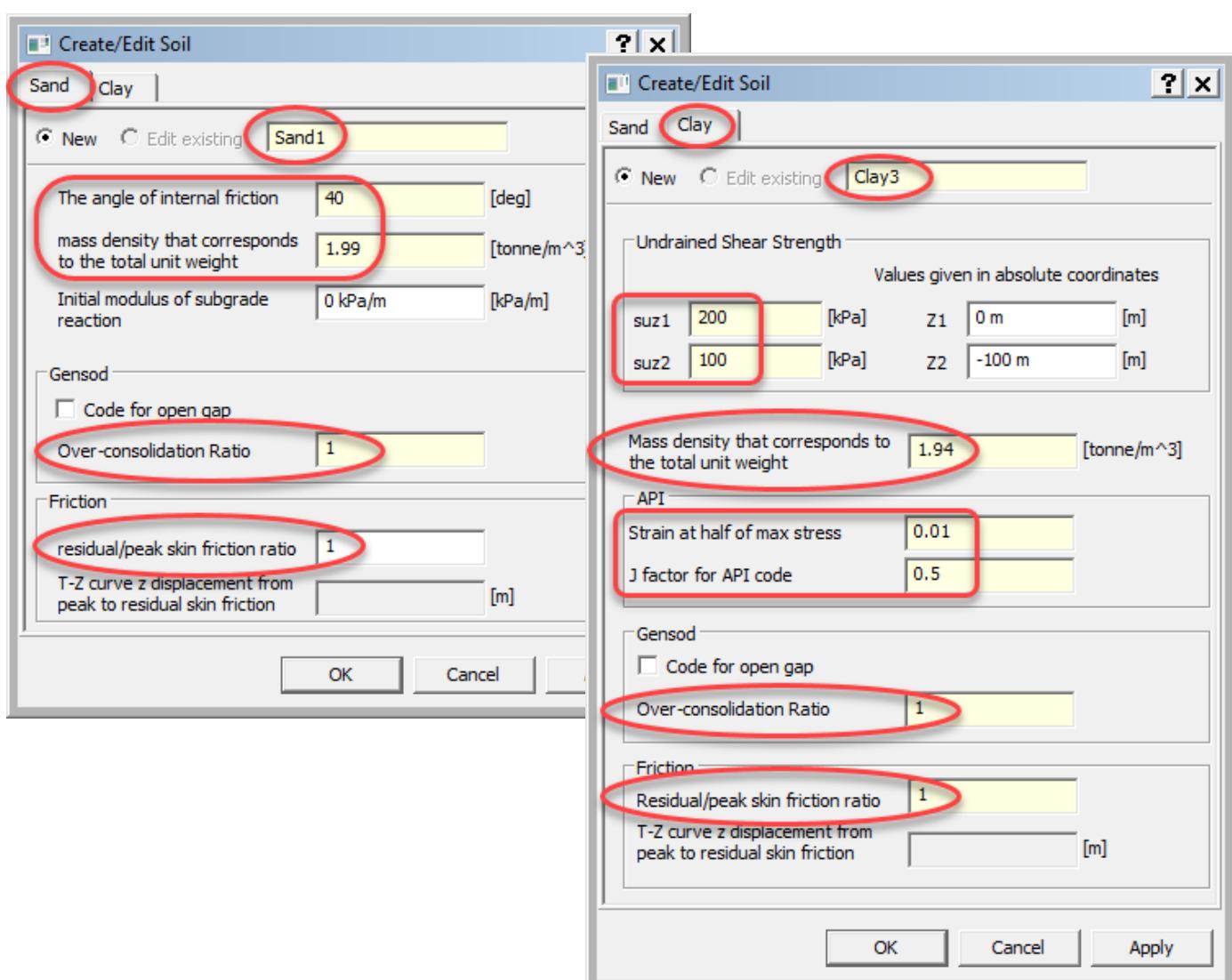
➤ Soil curves to be used are:

- p-y according to API 1987
- t-z according to API 1993
- q-z according to API 1993

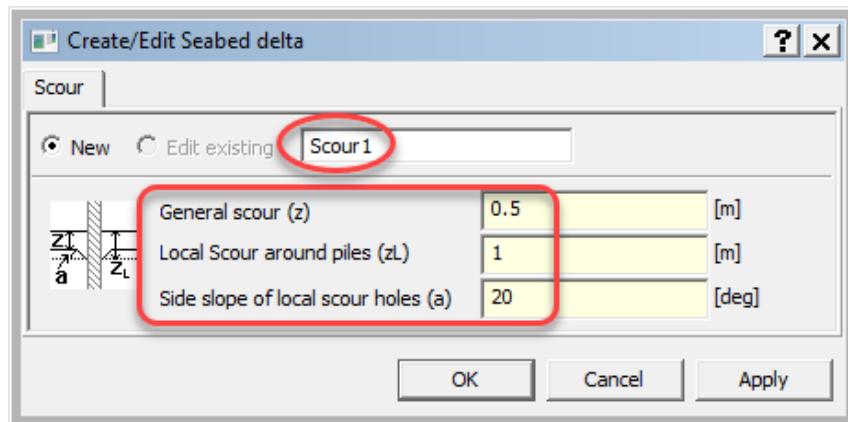
➤ Scour (erosion) data:

- General scour (erosion) = 0.5 m
- Local scour around piles = 1 m
- Slope of local scour = 20 m

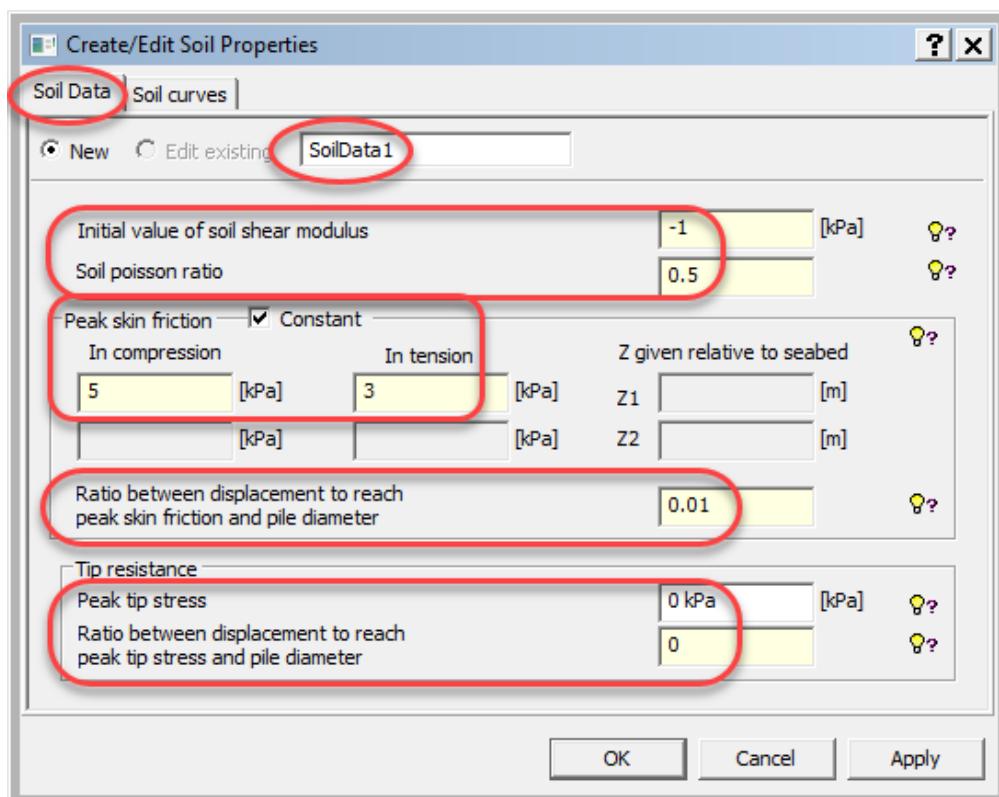
➤ Right-click the *Environment | Soil* folder in the browser and select *New Sand/Clay* to enter data for sand/clay layers. Below is shown the dialog for the layers named Sand1 and Clay3. Enter data for the three sand layers and the two clay layers.



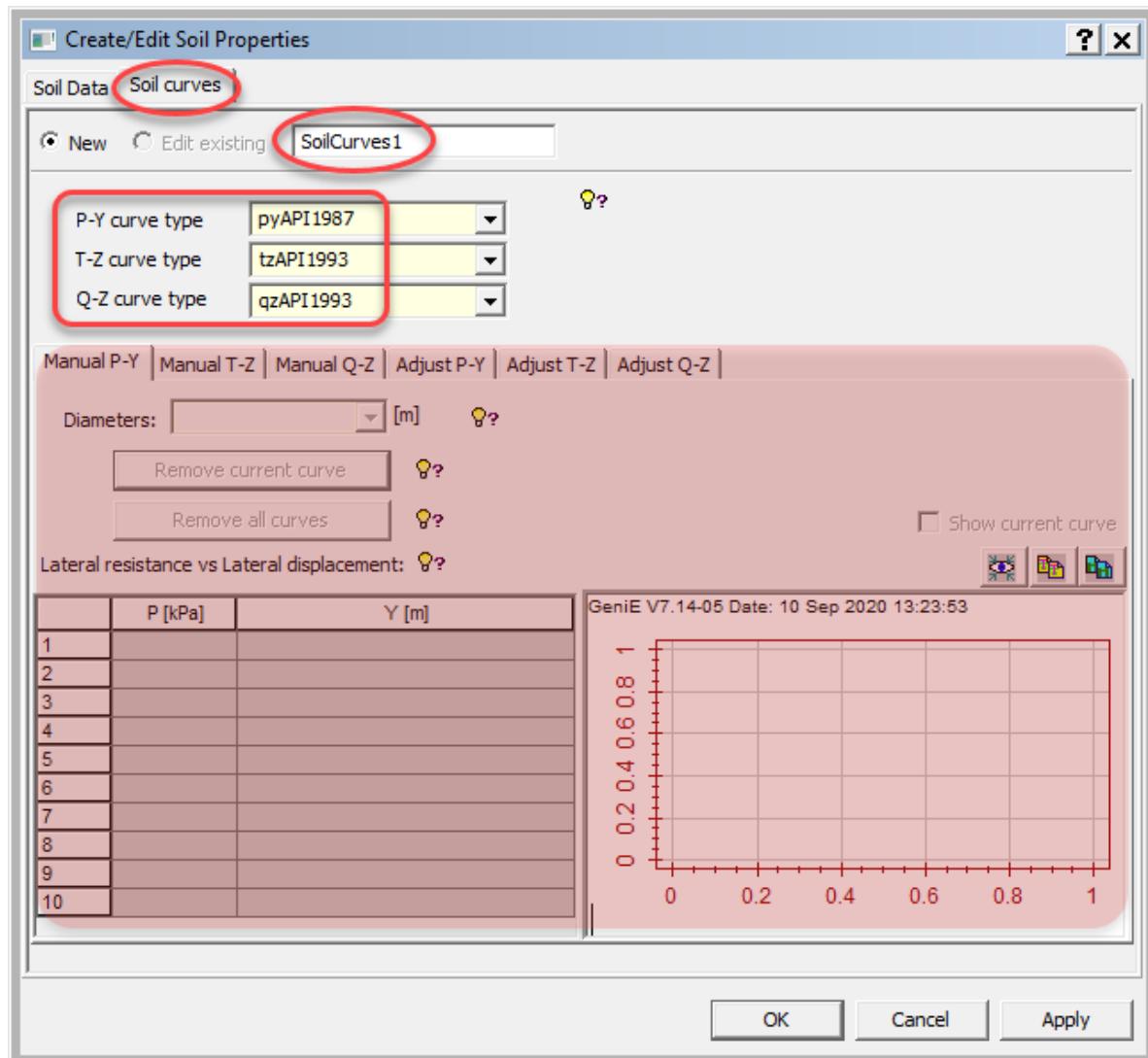
- Right-click the *Environment | Soil* folder in the browser and select *New Scour* to enter scour (erosion) data as shown below.



- Right-click the *Environment | Soil* folder in the browser and select *New Soil Data* to enter additional soil data for the soil layers. Below is shown the dialog for *SoilData1*. Enter data for all six soil data.

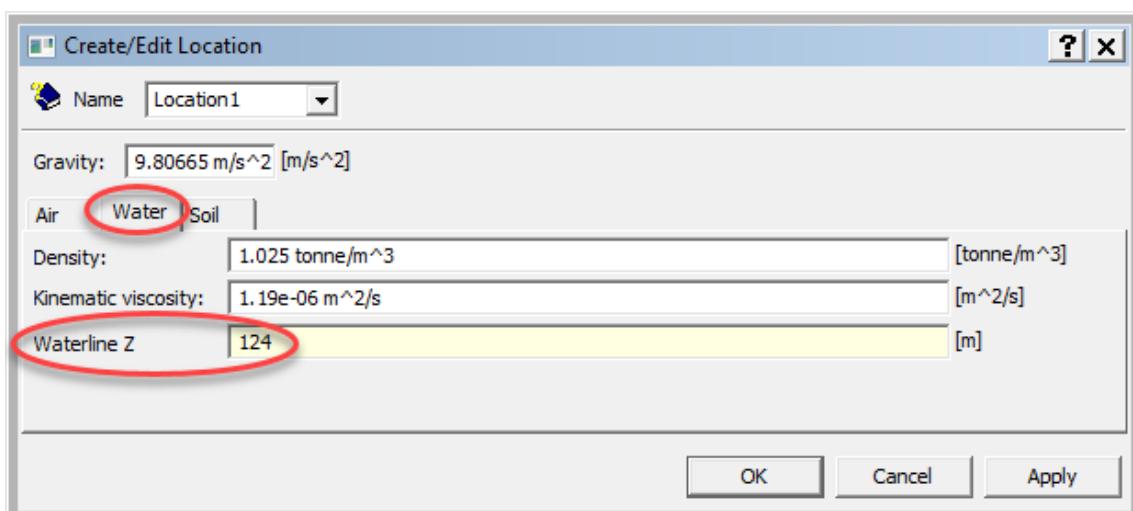


- Right-click the *Environment | Soil* folder in the browser and select *New Soil Curves* to select soil curves as shown below. The area for manual specification of soil curves is not used in this case.

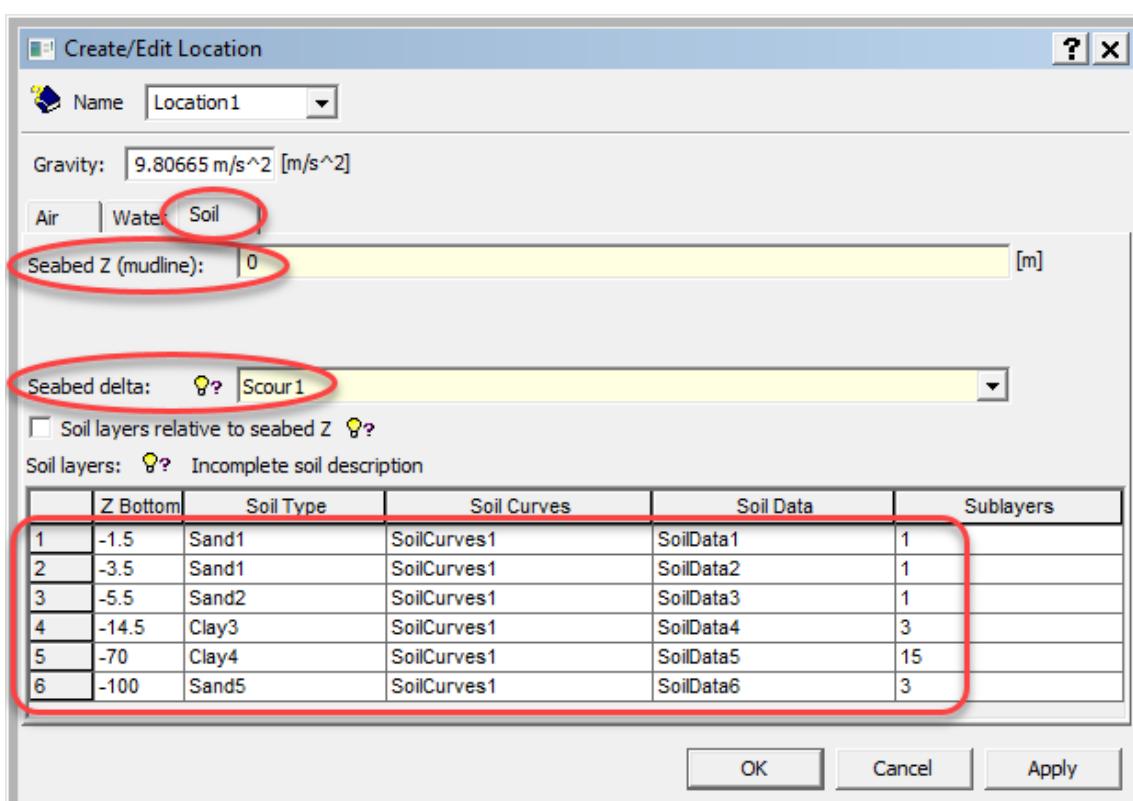


14 CREATE LOCATION

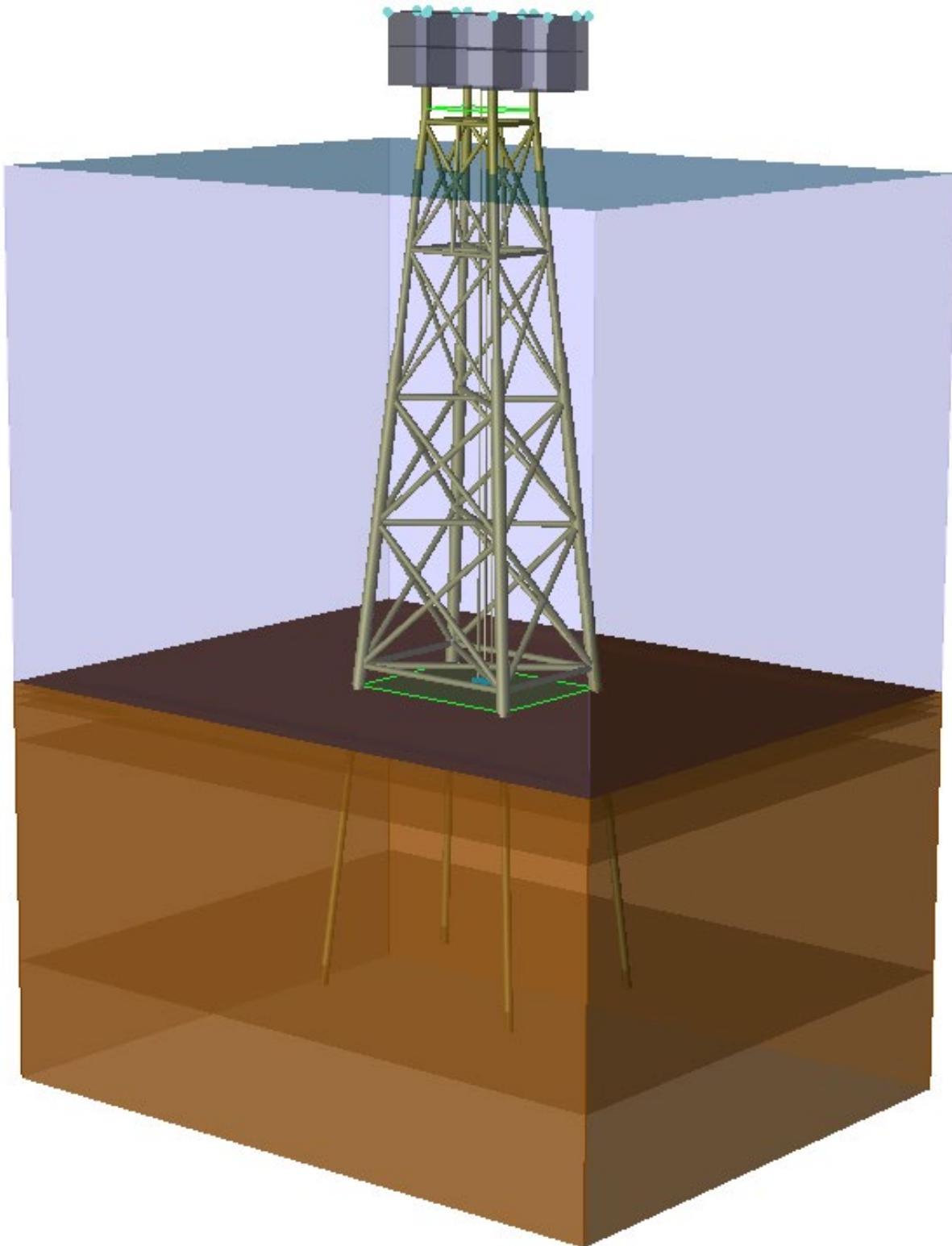
- Right-click the *Environment* folder in the browser and select *New Location* to create a so-called *location*. A location contains data about:
- Acceleration of gravity – accept the default value
 - Air density and kinematic viscosity – irrelevant in this exercise
 - Water density, kinematic viscosity and still water line Z-coordinate, enter data as shown below. I.e. the water depth is 124 m since the seabed is at 0 m.



- Soil: seabed Z-coordinate, scour information and data on soil layers, enter data as shown below.

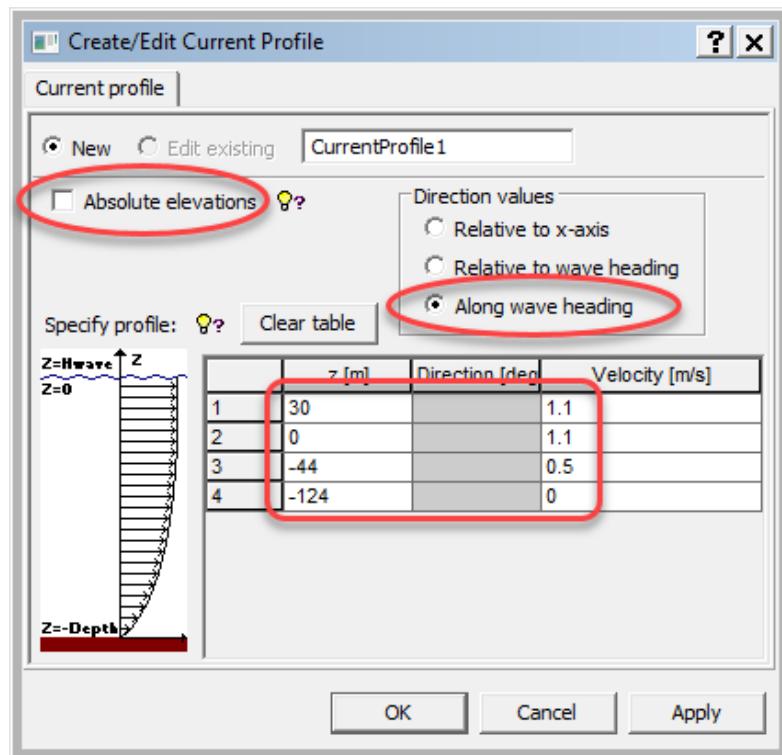


- Having defined the location switch to *Default display* configuration to see the jacket with piles in the water and soil environment.

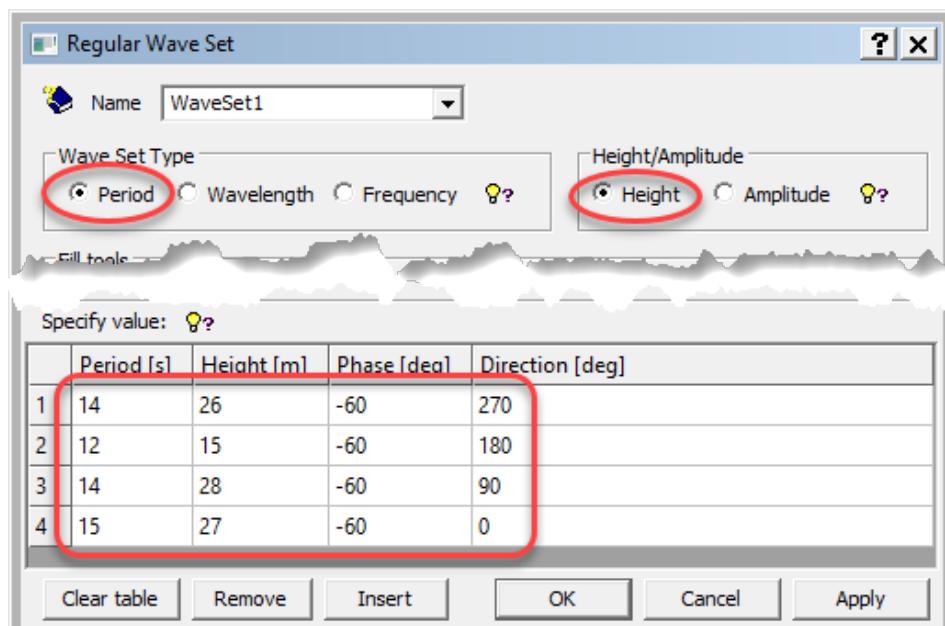


15 CREATE CURRENT AND WAVES

- Create a current profile by right-clicking the *Environment | Water* folder in the browser and selecting *New Current Profile*. Enter a current profile as shown below. Uncheck *Absolute elevations* to allow entering Z-coordinates relative to the still water line. The current is defined up to 30 m above the still water line to encompass all wave heights. Select *Along wave heading* so that the current always acts in the same direction as the waves.

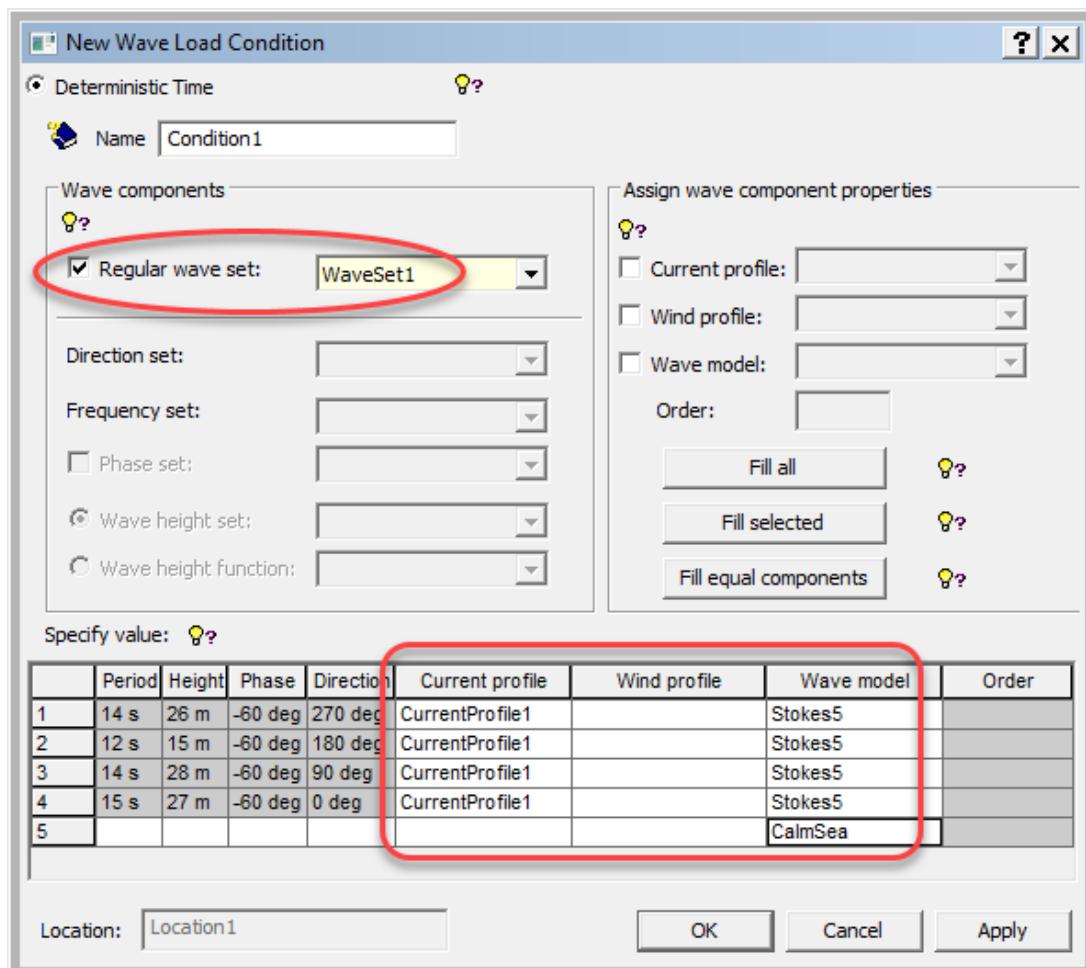


- Create a set of waves by right-clicking the *Environment | Water* folder in the browser and selecting *New Regular Wave Set*.



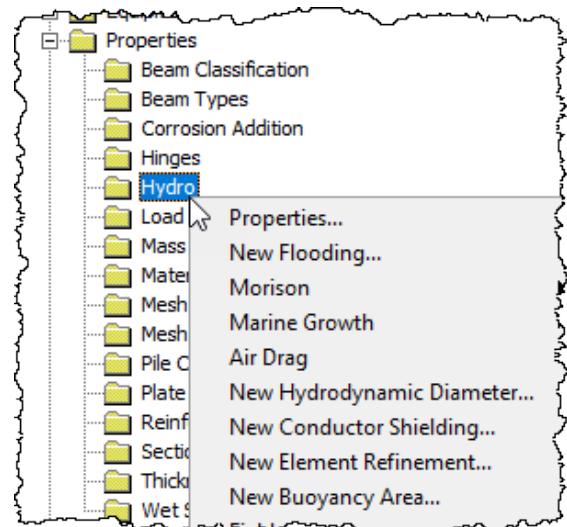
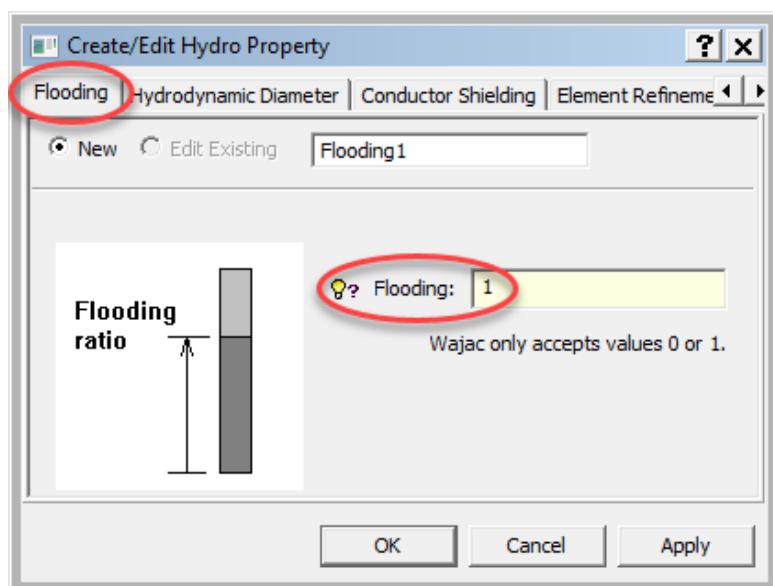
16 CREATE WAVE LOAD CONDITION

- A so-called *wave load condition* specifies wave theory to be used and also which current profile to be combined with the waves.
- Right-click the *Environment | Location* folder in the browser and select *New Wave Load Condition* to open the dialog below.
 - Pick up the wave set recently defined and see that the wave data fills the four leftmost columns.
 - Combine each wave with the recently defined current profile.
 - Select wave theory (wave model) Stokes 5th order to be used for all waves.
 - Add a 5th row to the table by selecting a *CalmSea* type of ‘*Wave model*’. This will be used as a buoyancy load case.

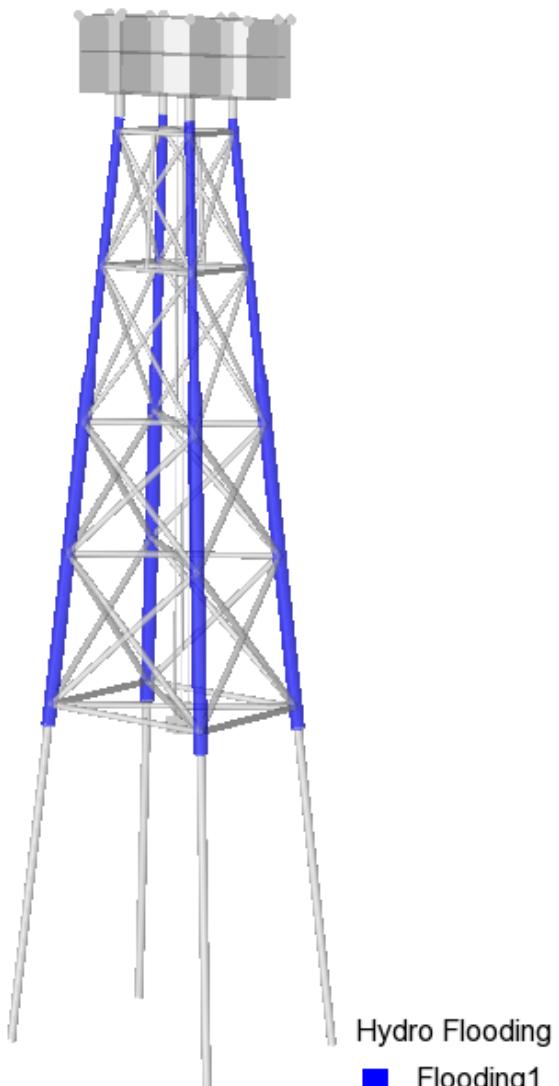


17 CREATE AND ASSIGN HYDRO PROPERTIES

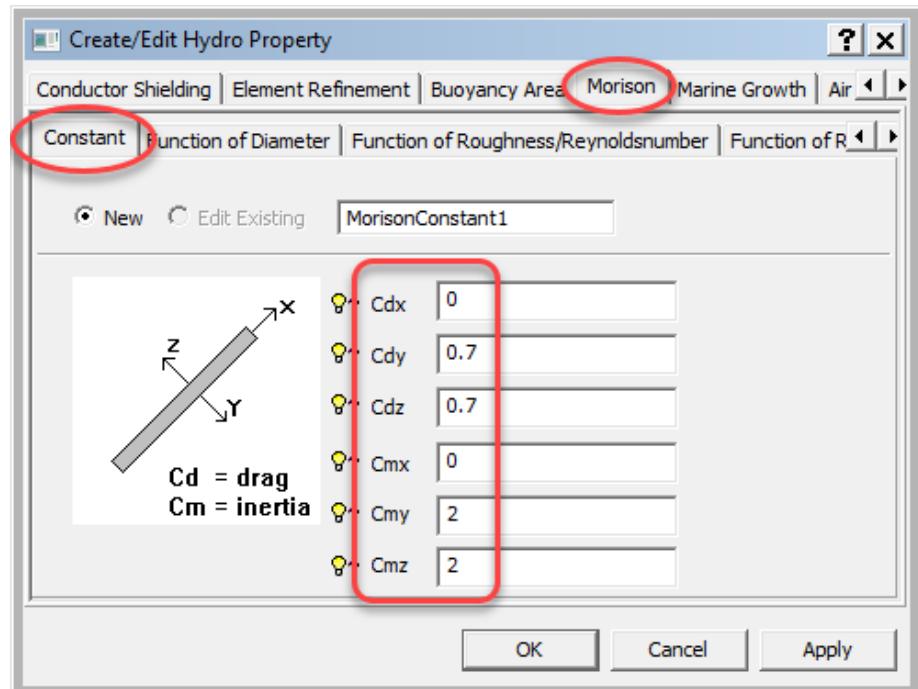
- The *Properties* folder in the browser includes a folder named *Hydro*. This stores data required for computing hydrostatic and hydrodynamic loads.
- Right-click the *Hydro* folder and select *New Flooding*. Give the value 1 which means fully flooded. (Values between 0 and 1 are currently not allowed, i.e. a tube cannot be partly flooded.)



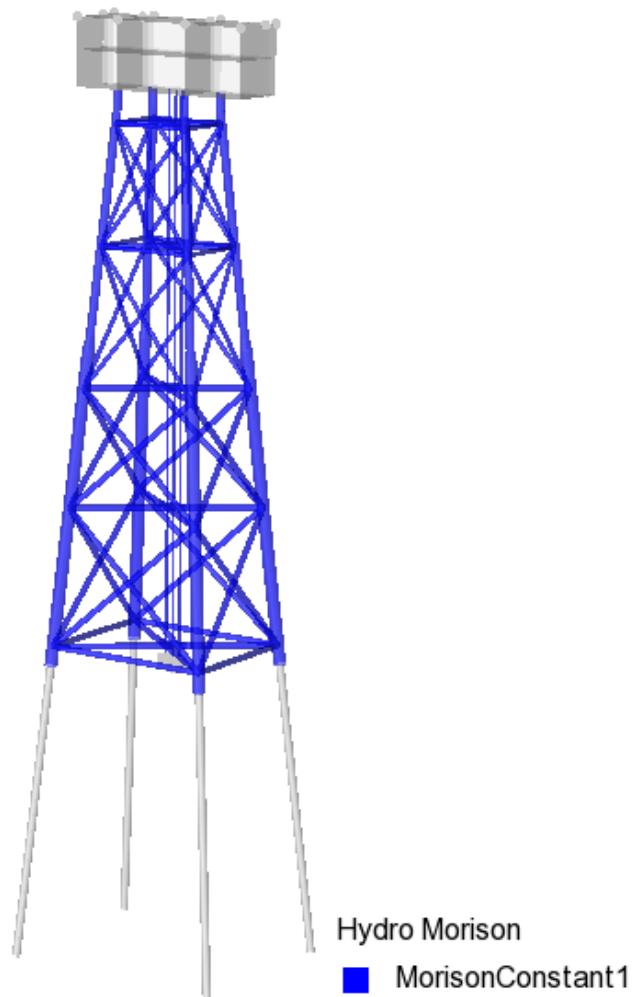
- Assign the flooding property to the legs. For this purpose use the set named Legs in the *Utilities | Sets | Regular Sets* folder.



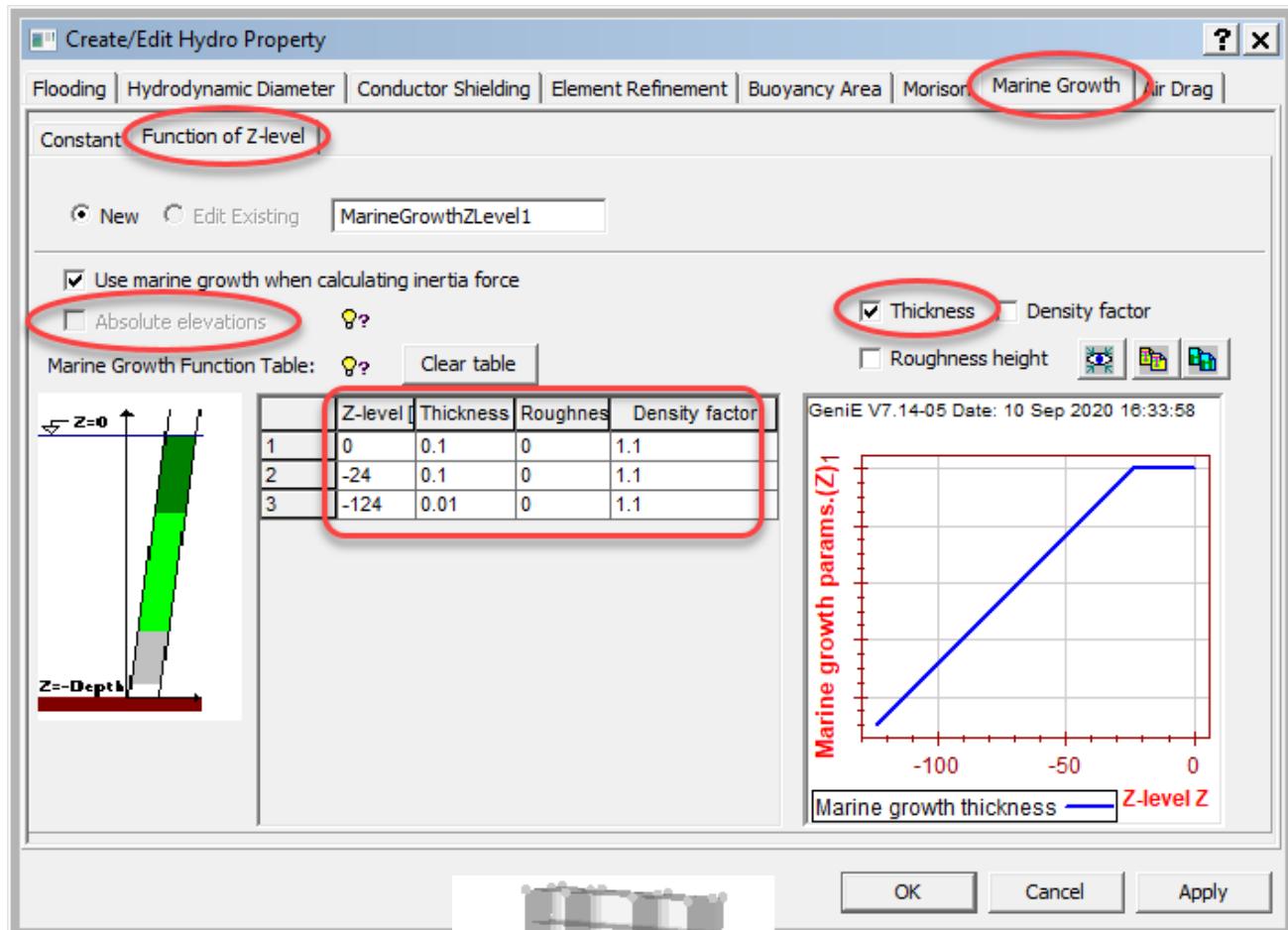
- Right-click the *Hydro* folder and select *Morison* | *New Morison Constant*. Accept the default data which means that the Morison drag coefficient is 0.7 and the Morison inertia coefficient is 2.



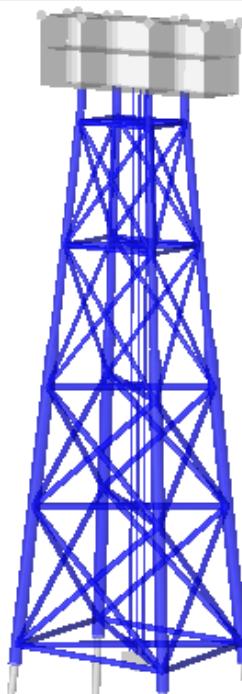
- Assign the Morison property to the jacket and the conductors. For this purpose use the sets named *Jacket* and *Conductors* in the *Utilities* | *Sets* | *Regular Sets* folder.



- Right-click the *Hydro* folder and select *Marine Growth | New Marine Growth Z-level*. Uncheck the *Absolute elevations* to allow entering Z-coordinates relative to the still water line. Enter data as shown below. The marine growth extends down to the sea floor (the depth is 124 m) as shown by the graph. Uncheck *Density factor* and *Roughness height* to include only *Thickness* in the graph.



- Assign the marine growth property to the jacket. For this purpose use the set named *Jacket* in the *Utilities | Sets | Regular Sets* folder.

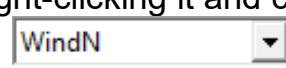


18 CREATE LOADS

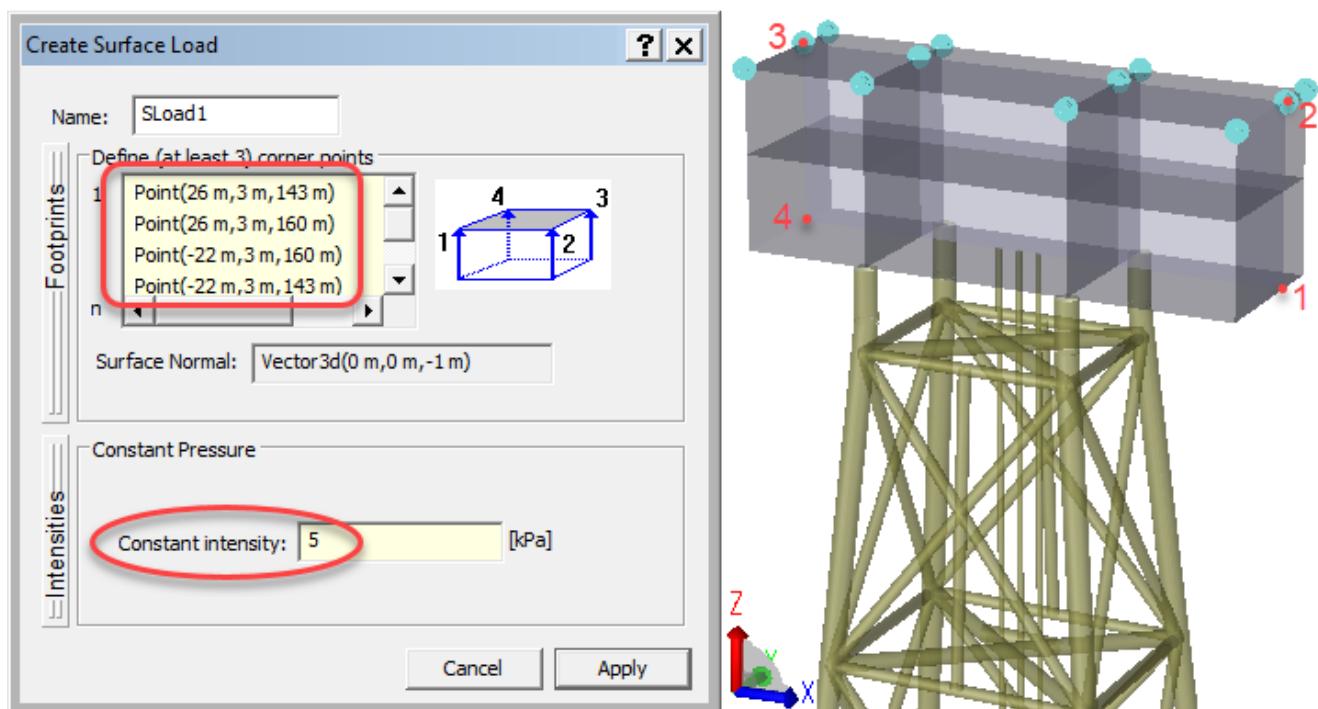
➤ Create the following load cases by *Loads | Load Case*:

1. Gravity (self weight)
2. WindN – static wind pressure of 5 kPa from north, i.e. in negative Y-direction
3. WindE – static wind pressure of 7 kPa from east , i.e. in negative X-direction
4. WindS – static wind pressure of 6 kPa from south, i.e. in positive Y-direction
5. WindW – static wind pressure of 9 kPa from west, i.e. in positive X-direction

➤ The Gravity load case is completed by right-clicking it, selecting *Properties* and checking *Include structure self-weight in structural analysis*.

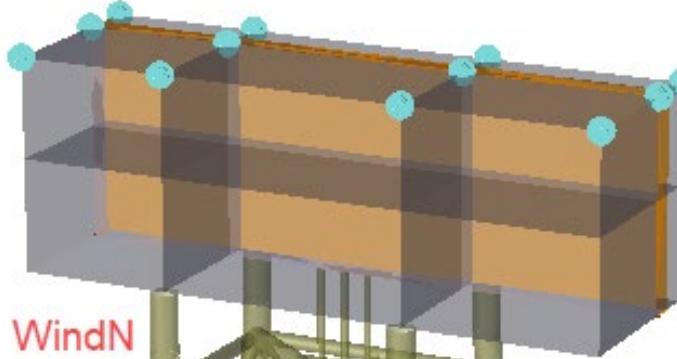
➤ Set the load case WindN as the currently selected by right-clicking it and clicking *Set Current*. Or select WindN in the load case selector: 

- Use *Loads | Explicit Load | Surface Load* to open the *Create Surface Load* dialog.
- Click in sequence the four points as indicated in the model, i.e. the four corners of the vertical wall in the XZ-plane, to position the pressure load.
- Give *Constant Pressure*, *Constant intensity* of 5 kPa. Note the sketch in the dialog telling direction of positive pressure.

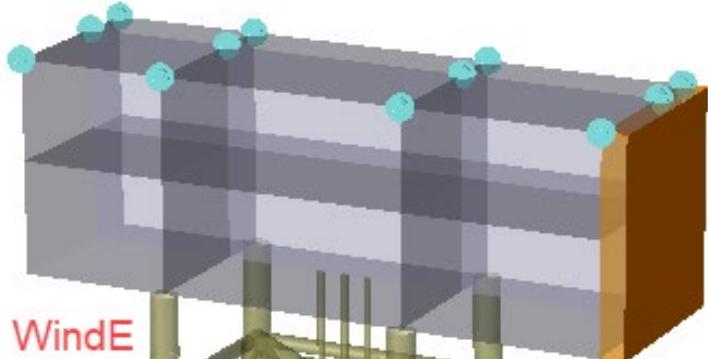


➤ Repeat the process for the other three wind load cases. For the east and west wind loads select the surfaces closest to the wind.

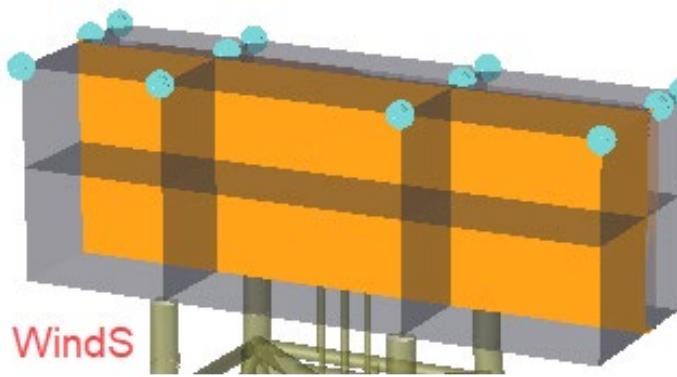
- The wind loads are displayed below.



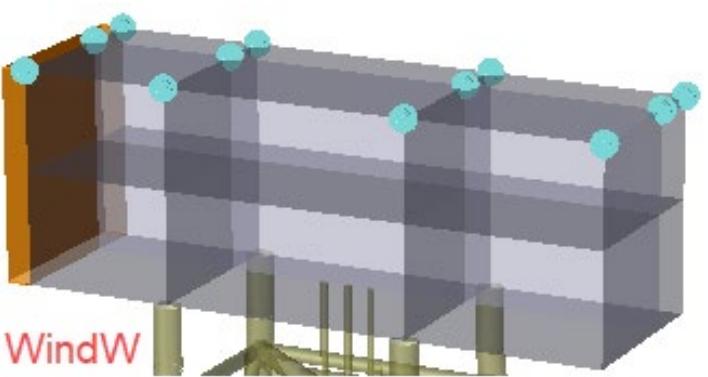
WindN



WindE

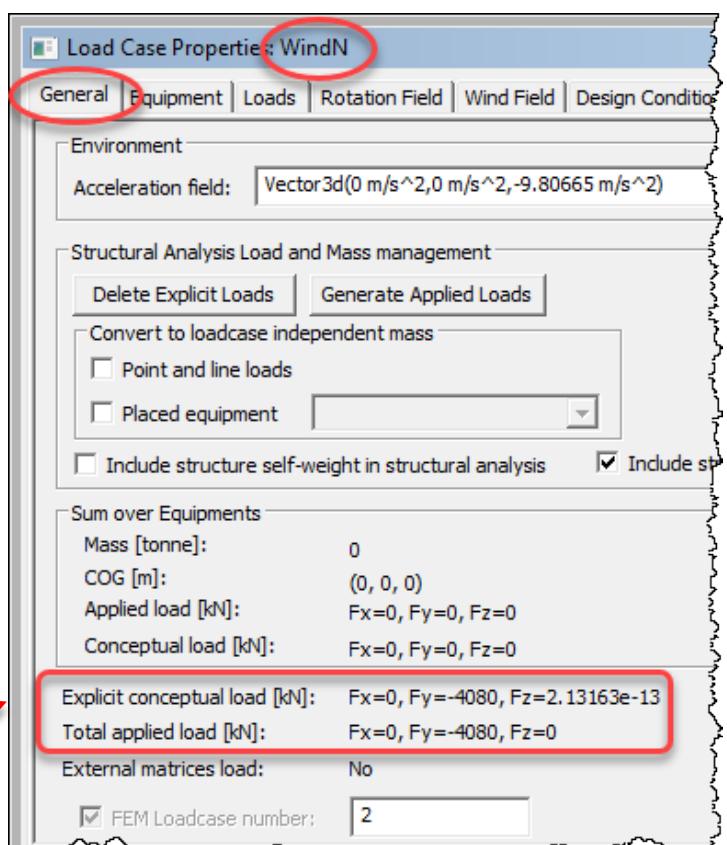
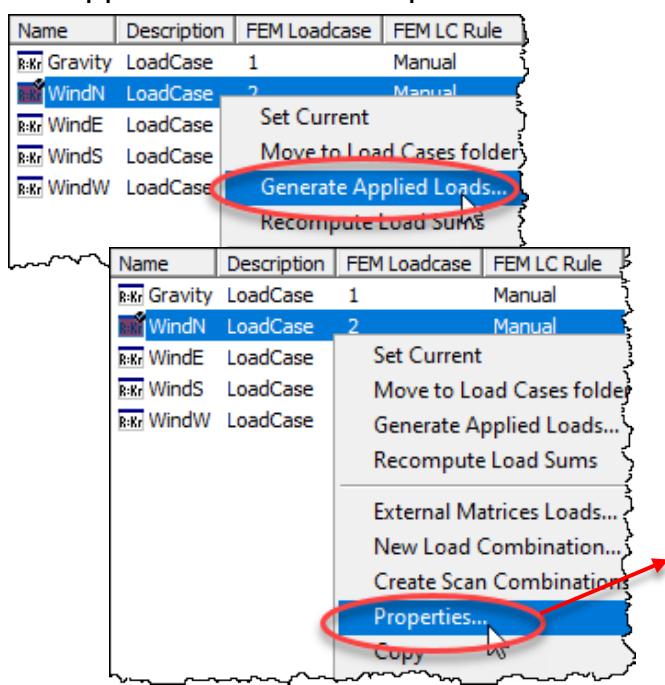


WindS



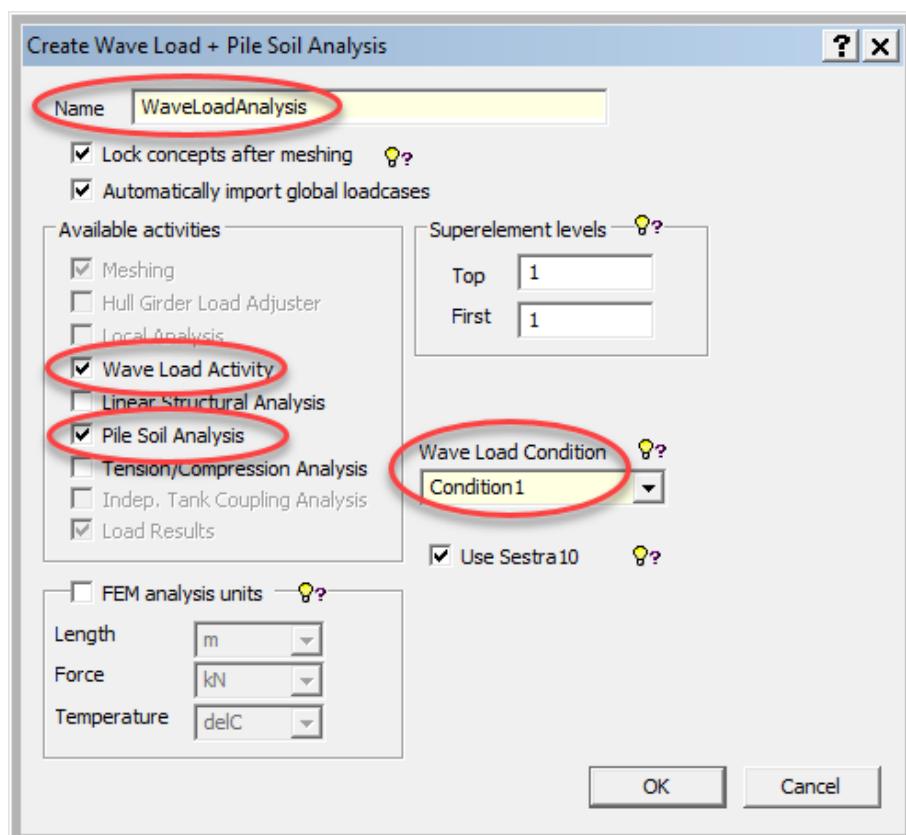
WindW

- To verify a load right-click it and select *Generate Applied Loads* followed by right-clicking it once more and selecting *Properties*. In the *Load Case Properties* dialog find the *Explicit conceptual load* – the user-specified load, and the *Total applied load* – the load as applied to beams and plates.

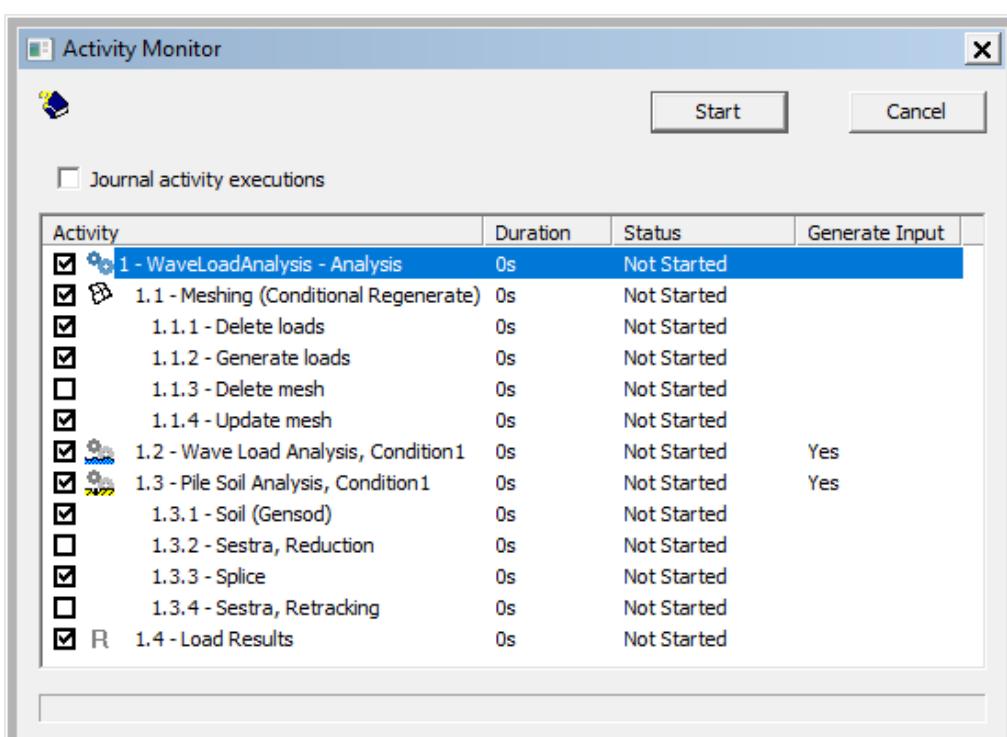


19 CREATE WAVE LOAD ANALYSIS ACTIVITY

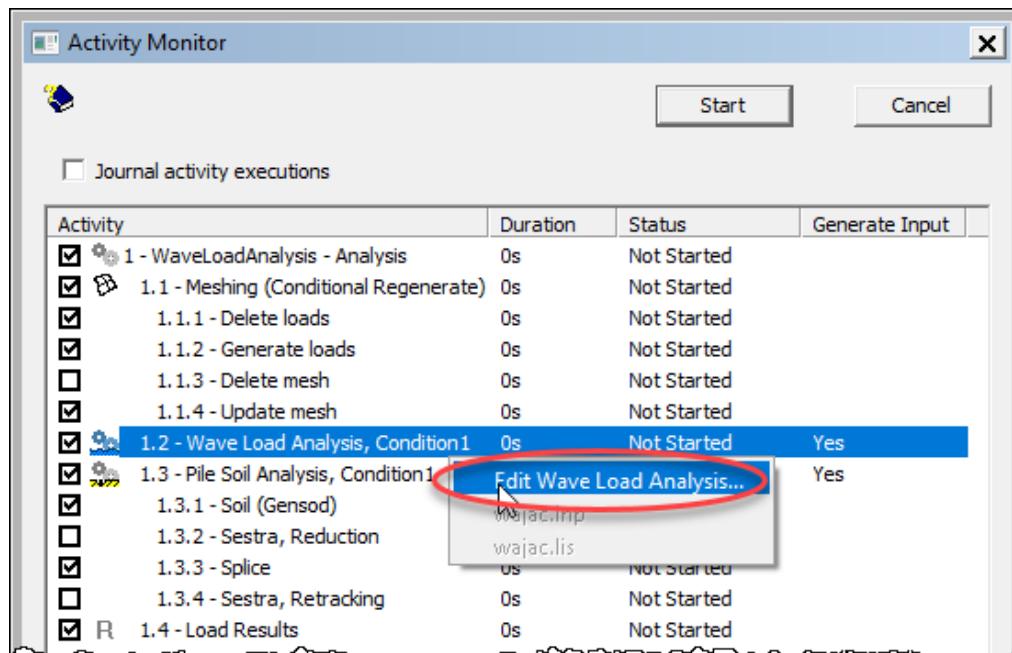
- Use *Mesh & Analysis | Activity Monitor* (or Alt+D) to open the dialog below. Select *Wave Load Activity* and *Pile Soil Analysis*. In the *Wave Load Condition* pulldown menu select the previously created wave load condition.



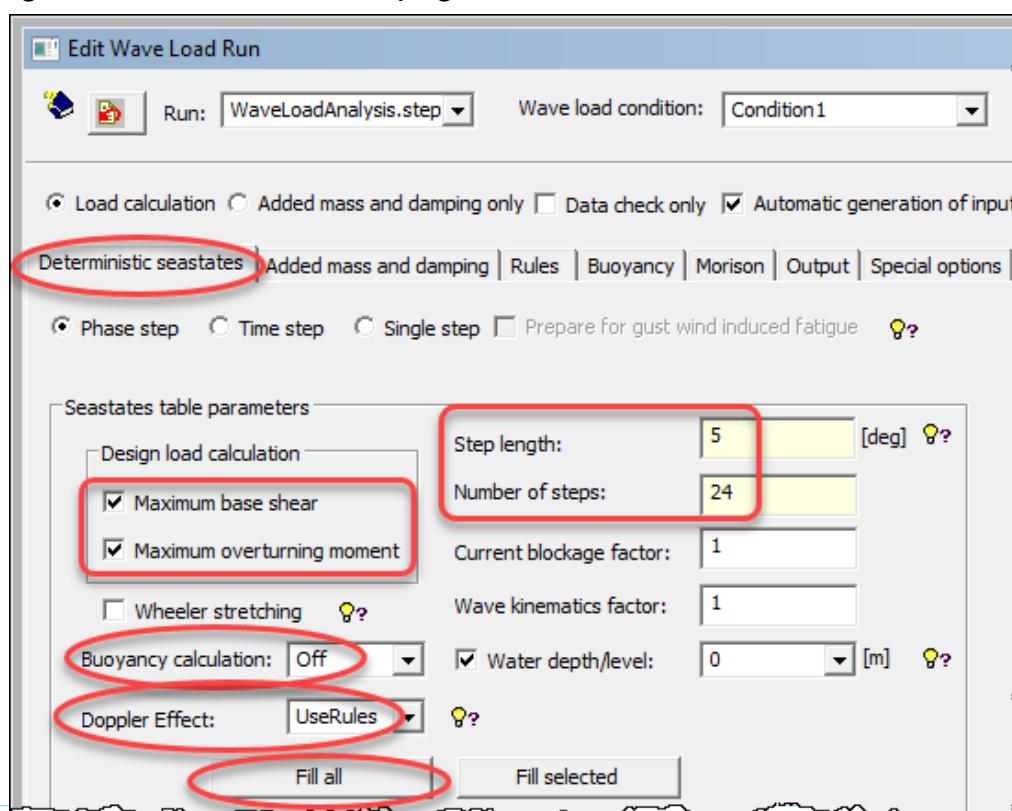
- The *Activity Monitor* opens up. But the analysis is not yet ready to be run.



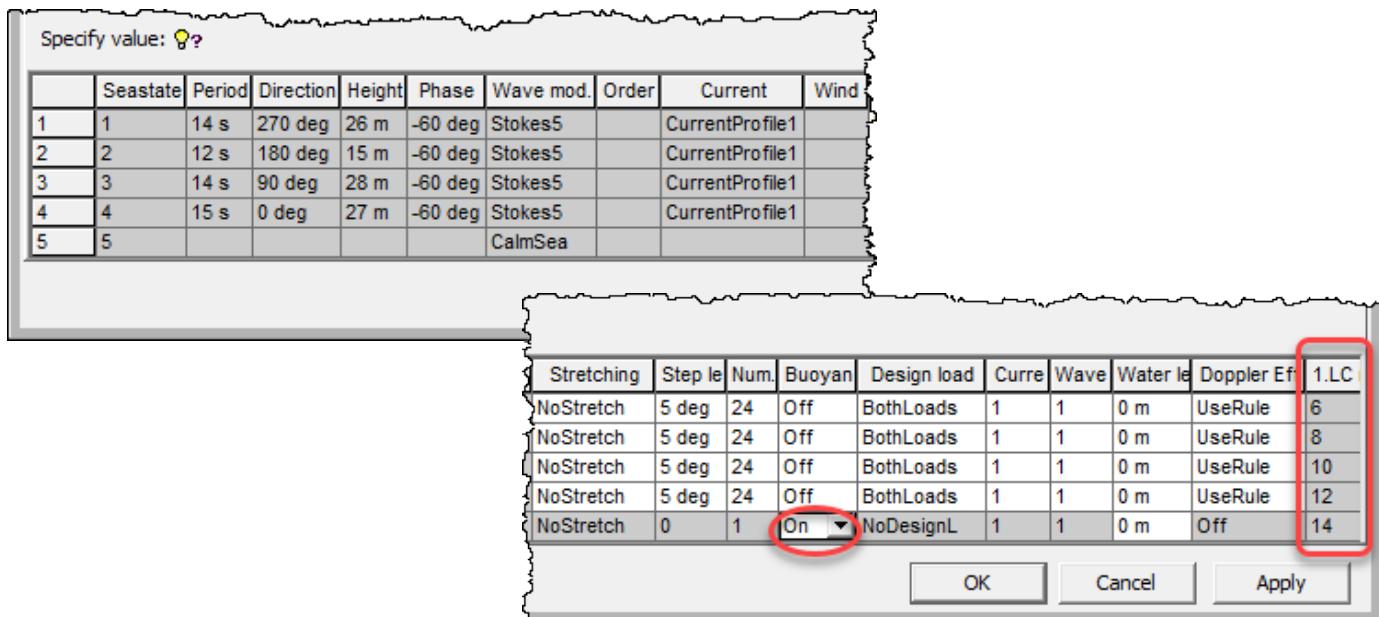
- Right-click the *Wave Load Analysis* activity and select *Edit Wave Load Analysis*.



- In the *Edit Wave Load Run* dialog, the *Deterministic seastates* tab, set *Step length* to 5 deg and *Number of steps* to 24, i.e. wave loads will be calculated for 24 steps of 5 deg starting at -60 deg (as specified for the wave set). Check *Maximum base shear* and *Maximum overturning moment* to have only wave loads for these two phase steps stored. Set *Buoyancy calculation* to *Off* and *Doppler Effect* to *UseRules*. With these settings, click *Fill all* and see that the table at the bottom of the dialog is filled as shown next page.



- The lower part of the *Edit Wave Load Run* dialog is filled with data as specified. For row 5, which is wave model *CalmSea*, change *Off* to *On* in the *Buoyancy* column. I.e. this is a pure buoyancy load case.



- When clicking *Apply*, the last column, *1.LC num.*, will be filled with numbers.
 - 6 in the first row means that the first load case pertaining to the first wave will be load case 6. This is because there are five manually created load cases: gravity and four wind loads.
 - There will be two load cases, 6 and 7, pertaining to the first wave, one for the *Maximum base shear* and one for the *Maximum overturning moment*.
 - 8 in the second row means that the first load case pertaining to the second wave will be load case 8, i.e. subsequent to the two load cases pertaining to the first wave.
 - And so on for the next waves.
 - The buoyancy load case will be number 14.
- Click *OK* to close the *Edit Wave Load Run* dialog but do not start the analysis yet. Load combinations must be created first to establish real loading situations. So close the *Activity Monitor*.
 - Note that due to the non-linear pile-soil analysis combinations cannot be created subsequent to the analysis – so-called smart load combinations in GeniE – as such presuppose linearity.

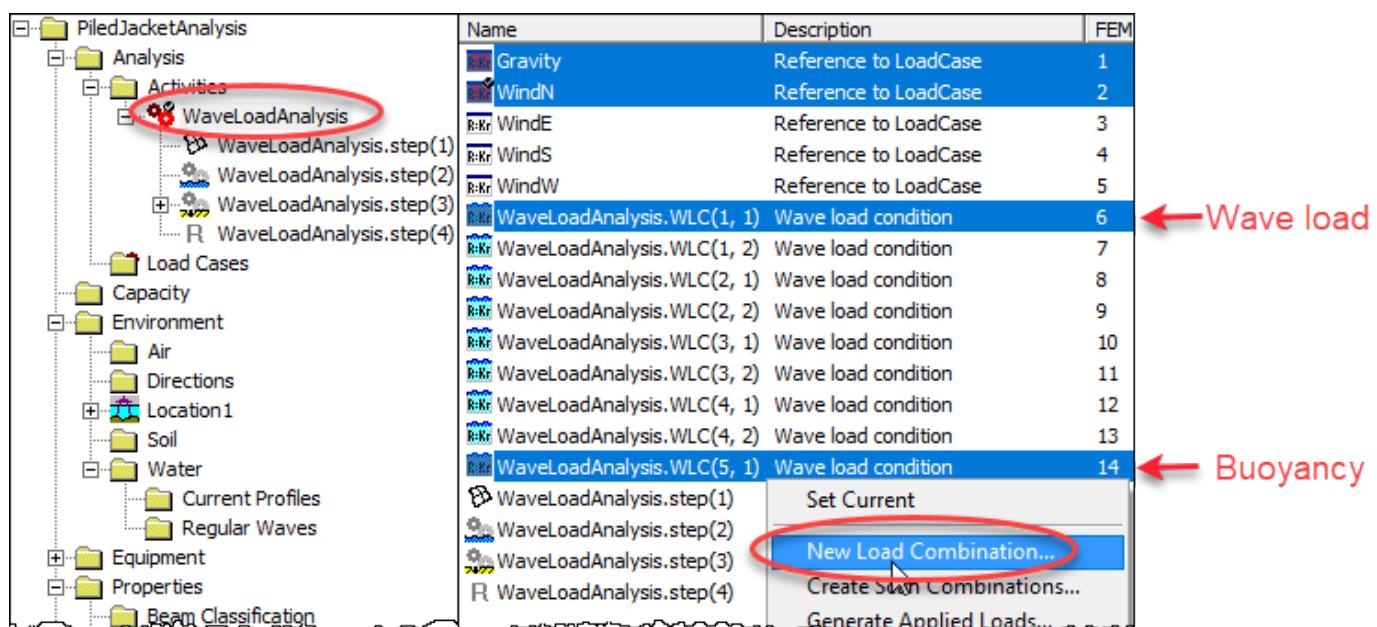
20 CREATE LOAD COMBINATIONS

➤ Create the eight load combinations presented in the table below.

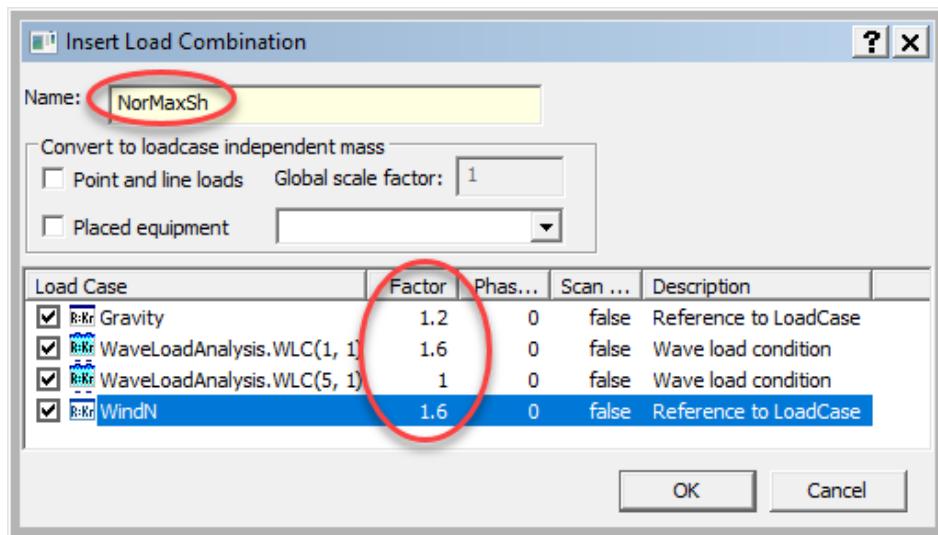
- Note that the table below is based on the assumption that maximum base shear is the first and maximum overturning moment is the second load case stored for each wave. This will be the case if maximum base shear occurs prior to maximum overturning moment when the wave travels through the structure.

Load combination	Wave load WLC(n,n)	Load factor	Buoyancy load WLC(n,n)	Load factor	Gravity load	Load factor	Wind load	Load factor
NorMaxSh	1,1	1.6	5,1	1.0	Gravity	1.2	WindN	1.6
NorMaxMo	1,2	1.6	5,1	1.0	Gravity	1.2	WindN	1.6
EasMaxSh	2,1	1.6	5,1	1.0	Gravity	1.2	WindE	1.6
EasMaxMo	2,2	1.6	5,1	1.0	Gravity	1.2	WindE	1.6
SouMaxSh	3,1	1.6	5,1	1.0	Gravity	1.2	WindS	1.6
SouMaxMo	3,2	1.6	5,1	1.0	Gravity	1.2	WindS	1.6
WesMaxSh	4,1	1.6	5,1	1.0	Gravity	1.2	WindW	1.6
WesMaxMo	4,2	1.6	5,1	1.0	Gravity	1.2	WindW	1.6

➤ The load combinations are most easily created by selecting the appropriate load cases appearing in the *Analysis | Activities | <name of analysis activity>* folder, right-clicking and clicking *New Load Combination*.



- In the *Insert Load Combination* dialog double-click in the *Factor* column to give the appropriate load case factor.

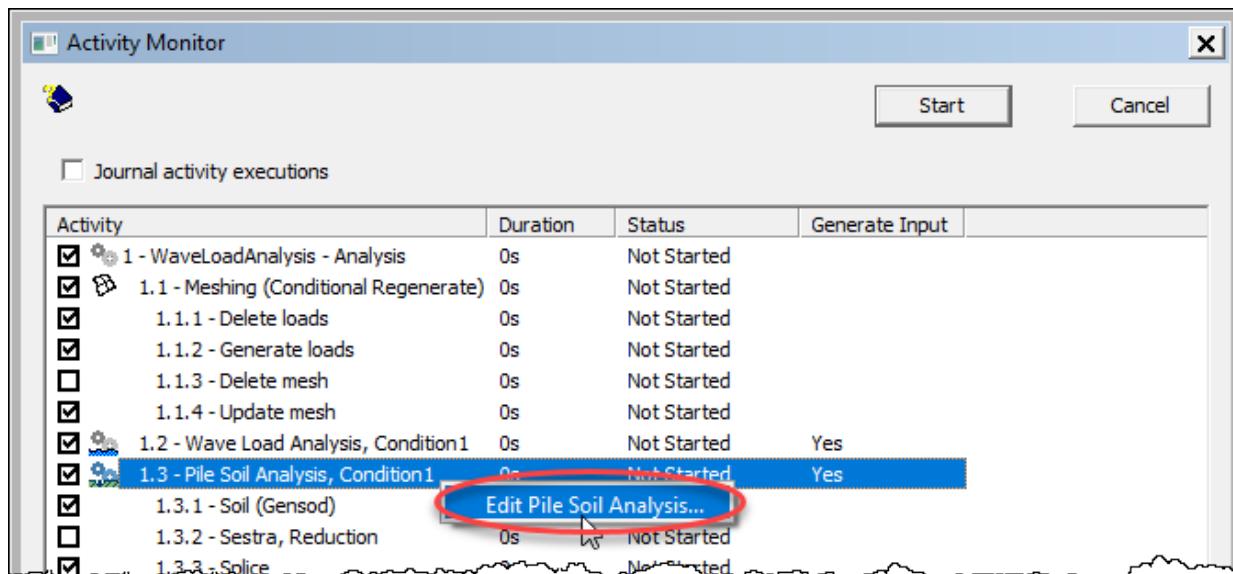


- Rather than creating all eight load combinations through the dialog above you may find it more convenient to create them through the *Command Line* interface.
- Do so by creating only the first combination, NorMaxSh, copy the commands for creating this load combination from the *Command Line* interface into an editor, edit this into creating the other seven combinations, and finally paste this back into the *Command Line* interface.
- These are the commands required for creating the load combination NorMaxSh:

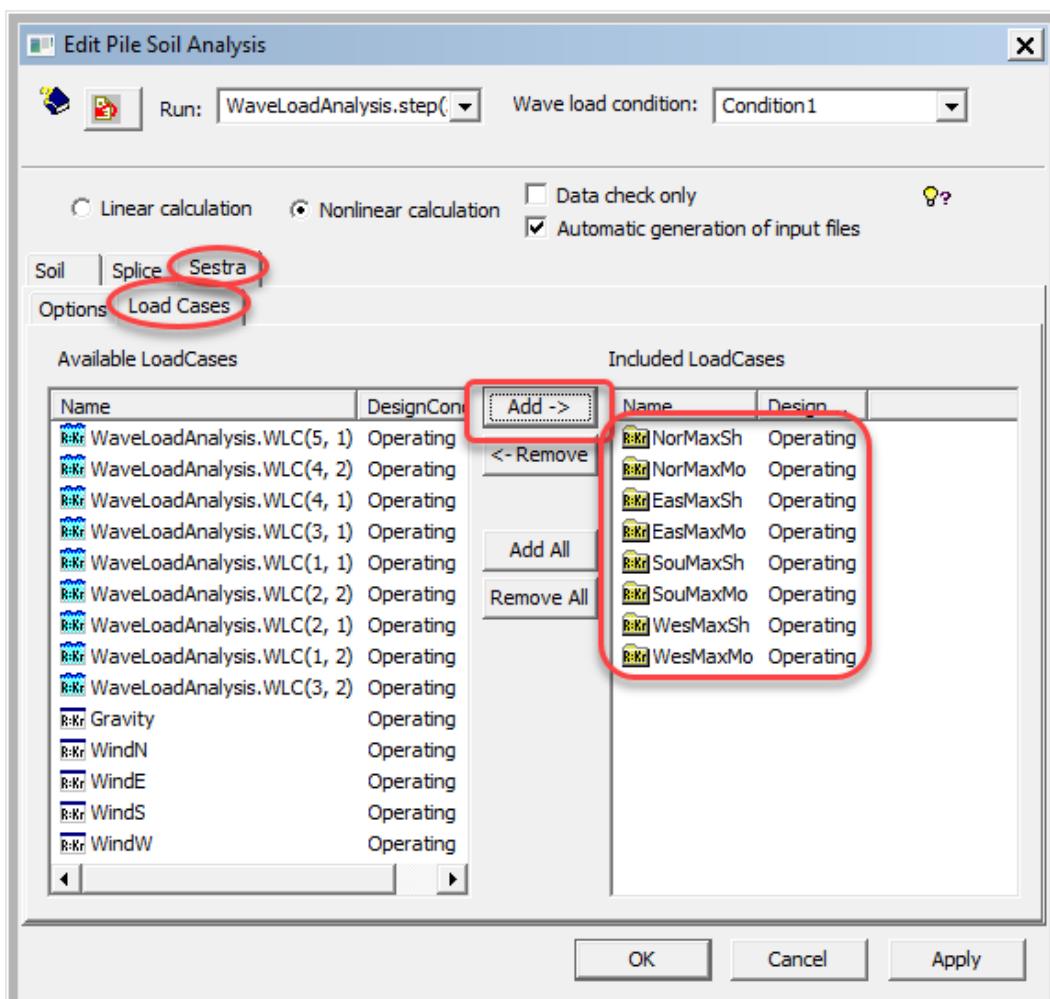

```
NorMaxSh = LoadCombination(WaveLoadAnalysis);
NorMaxSh.addCase(Gravity, 1.2);
NorMaxSh.addCase(WindN, 1.6);
NorMaxSh.addCase(WaveLoadAnalysis.WLC(1, 1), 1.6);
NorMaxSh.addCase(WaveLoadAnalysis.WLC(5, 1), 1);
```
- The following two commands are irrelevant in this case and can be skipped:


```
NorMaxSh.convertLoadToMass = false;
NorMaxSh.EquipmentRep = EquipmentAsLineLoads;
```

- Select the load combinations only for analysis. Do so by opening the *Activity Monitor* (Alt+D), right-clicking the *Pile Soil Analysis* and selecting *Edit Pile Soil Analysis*.

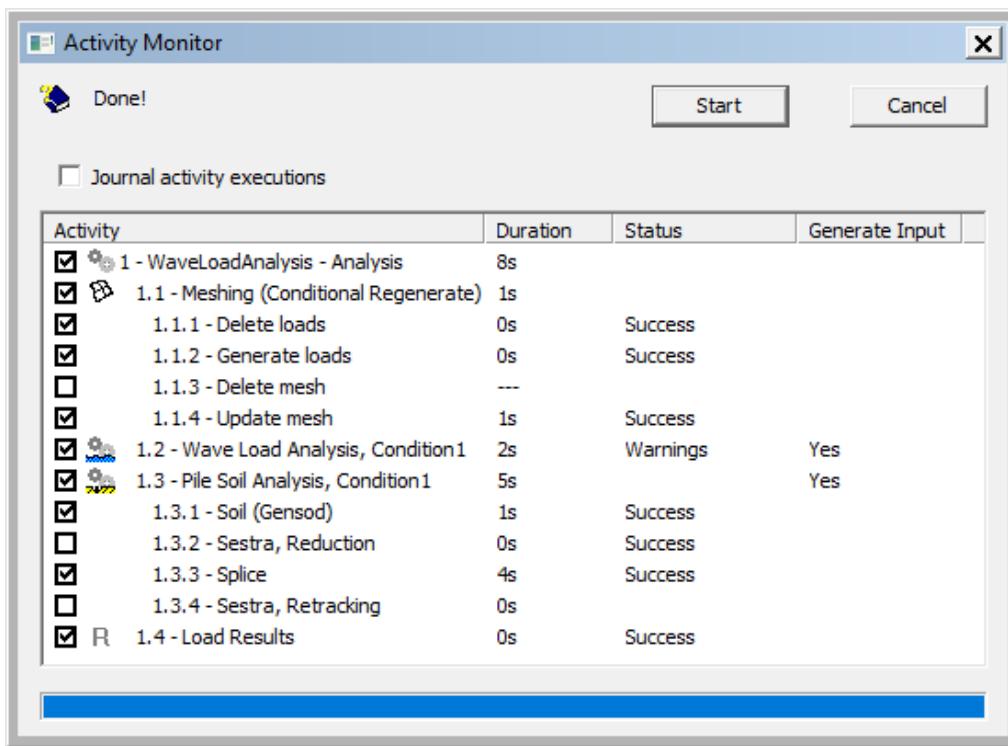


- Move the load combinations into the *Included LoadCases* field as shown below.



21 RUN WAVE LOAD ANALYSIS ACTIVITY

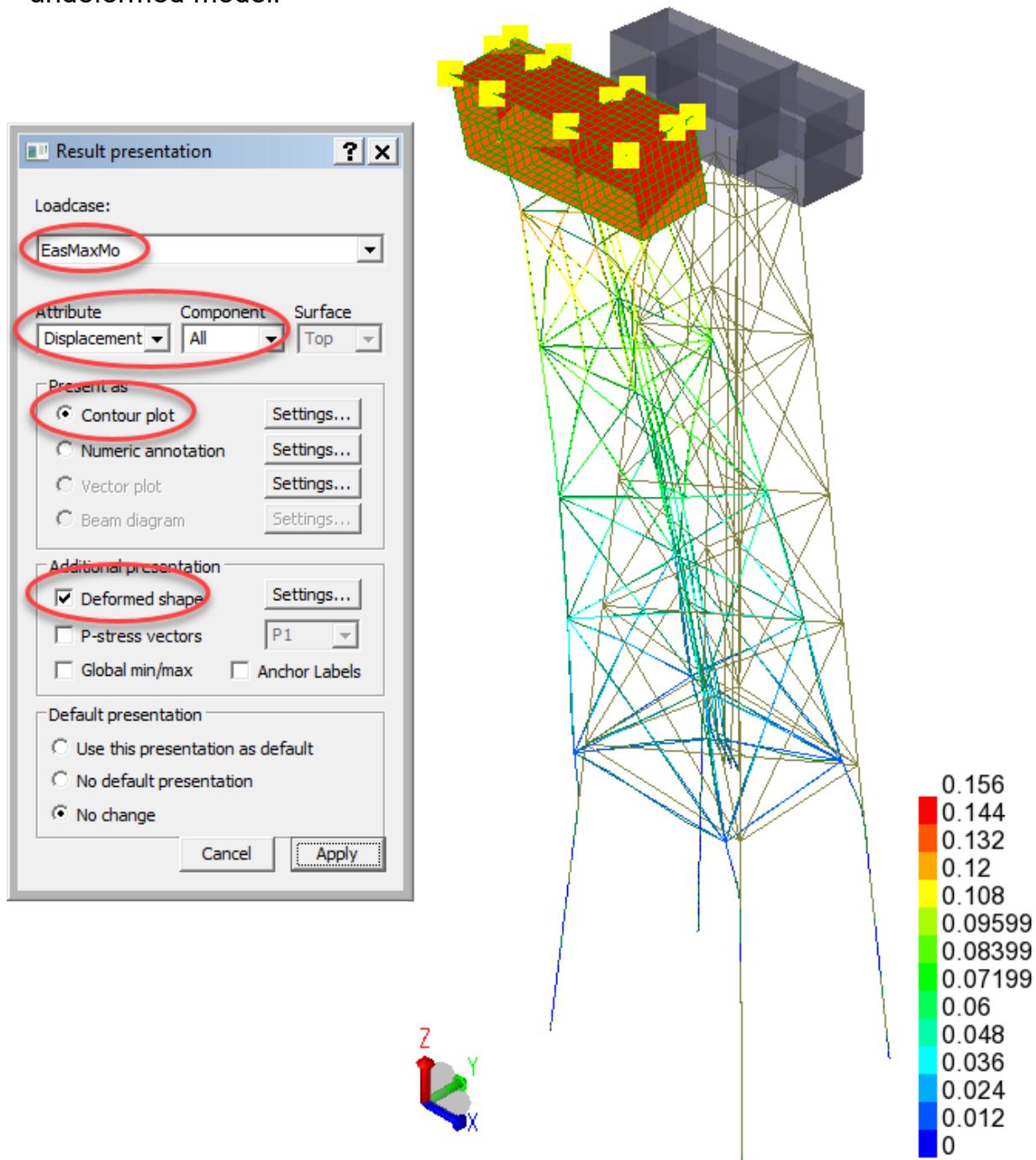
- In the *Activity Monitor*, click *Start* to run the analysis.



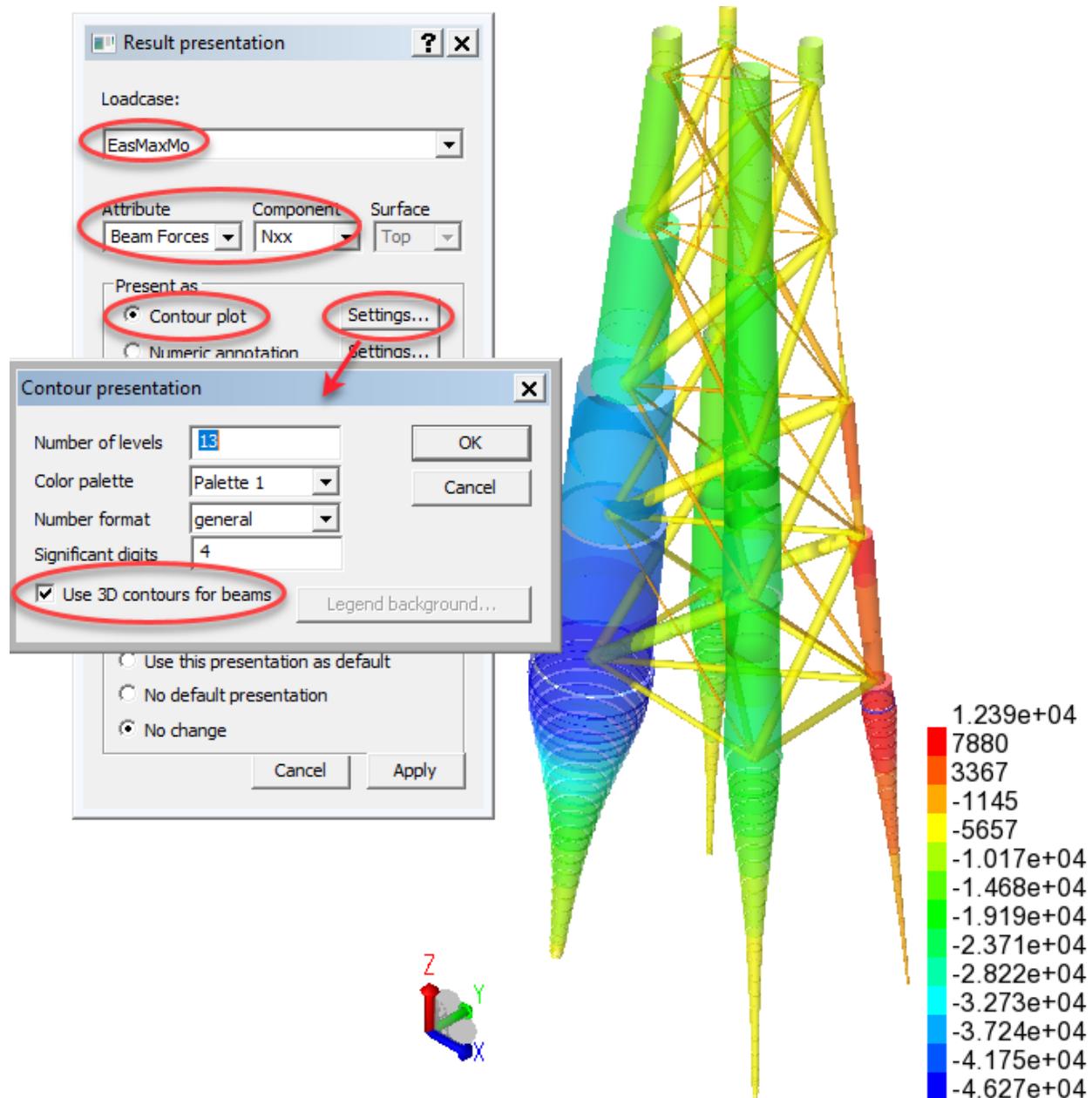
- The analysis runs the following batch programs in the background:
 - Wajac – wave load calculation
 - Sestra – linear structural analysis
 - Gensod (part of Splice) – generation of non-linear springs along piles
 - Splice – non-linear pile-soil analysis
 - Ensure all sub-activities succeed.
- Right-click the *Wave Load Analysis* to open the *Wajac.lis* file and confirm that the warning for this activity is the following one.
- ```
*** WARNING *** LOADCASE NUMBERS WILL OVERLAP ON THE FEM INTERFACE FILE AND THE LOAD FILE
This warning can in most cases be neglected.
```
- Right-click the *Splice* activity and select *Splice.mlg* and *Splice.lis* to see messages and summary of results from the Splice execution, respectively.

## 22 PRESENT RESULTS

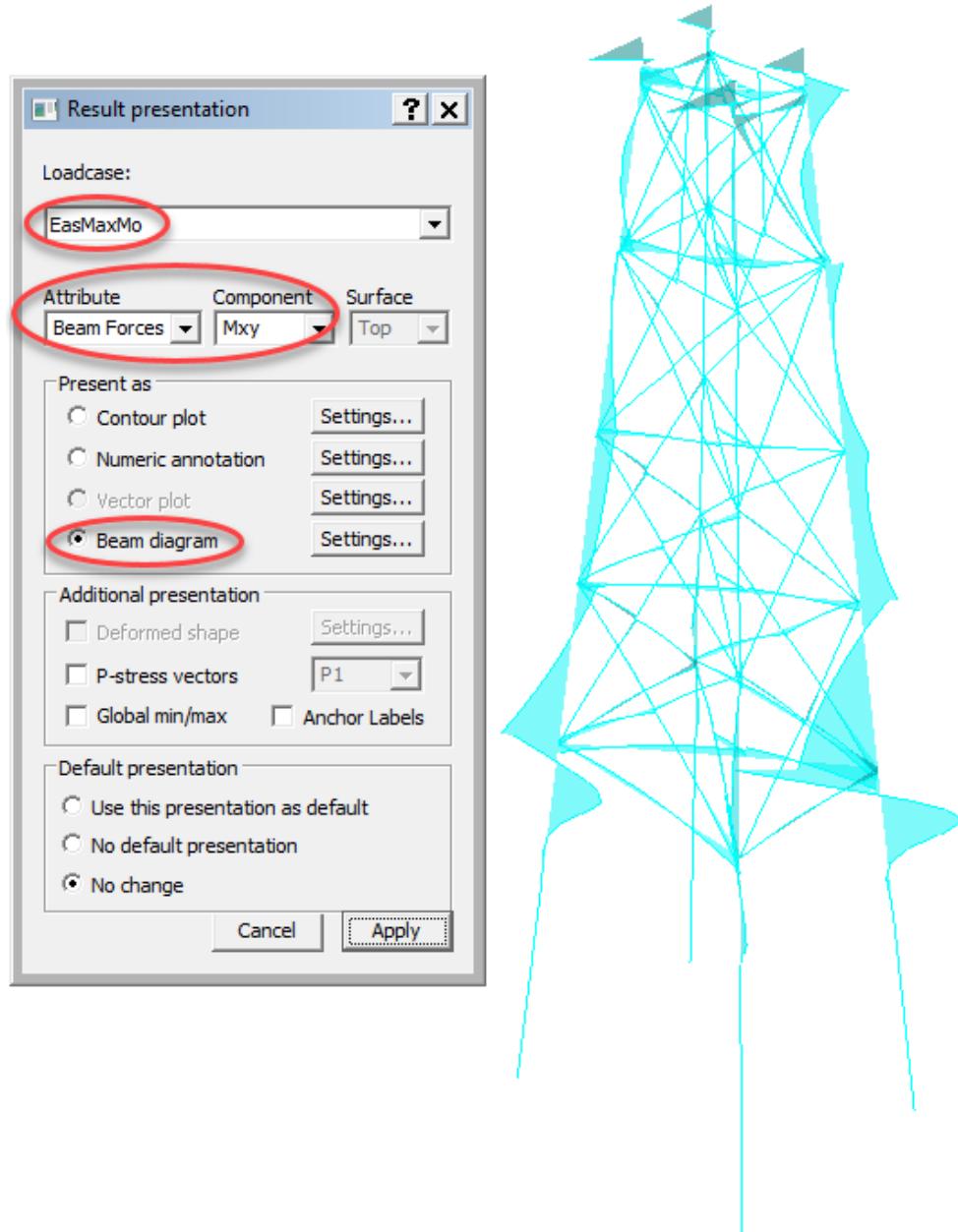
- Select *Results - All* display configuration to see some results.
  - Use *Results | Presentation* (or Alt+P) to open the *Result presentation* dialog. Select a load combination, an *Attribute* and *Component* and how to present the select result.
  - In the display below, the *Beam selection* button ( ) and *Plate selection* button ( ) have been right-clicked to 'open these eyes' so as to display the undeformed model.



- Another presentation is shown below: the axial beam forces with checking of *Use 3D contours for beams*.



- Yet a presentation is shown below: beam moment diagrams.



- A piled jacket analysis is normally concluded by code checking members and joints.  
Go to tutorial B9 for an exercise in code checking.

## 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.