SESAM TUTORIAL

GeniE
Wind Loads

Valid from program version 8.4

Sesam Tutorial

GeniE – Wind Loads

Date: September 2022

Valid from GeniE version 8.4

Prepared by: Digital Solutions at DNV

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

E-mail sales: digital@dvn.com

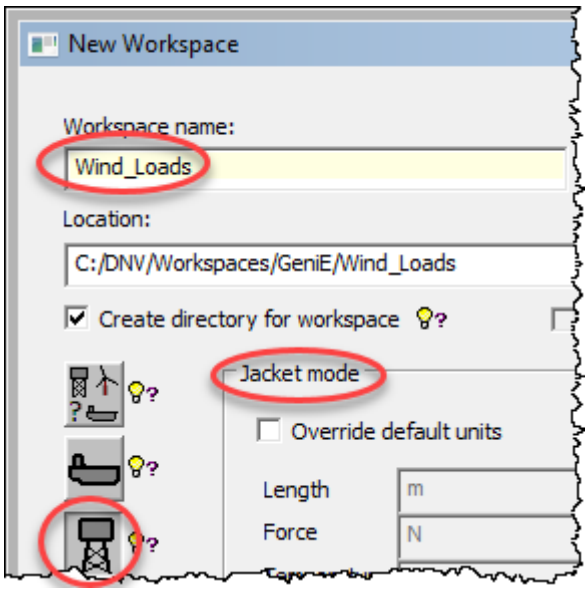
TABLE OF CONTENTS

1 INTRODUCTION

- This tutorial explains how to compute wind loads on a jacket and a topside structure with equipments, and how to combine the wind loads with wave loads on the jacket.
  - Wind loads on beams are calculated as line loads by Wajac using the Morison equation. Wajac is run in the background from GeniE.
  - Wind pressure on plates and equipments are calculated by GeniE. Wind pressures on equipments are converted into point and line loads on the structure supporting the equipments.

- This tutorial is based on importing a model of a jacket with a topside structure. Hence modelling of the jacket and topside structure is not included. For simplicity's sake, the jacket pile and soil foundation is not included in the imported model rather the jacket is fixed at the seabed.

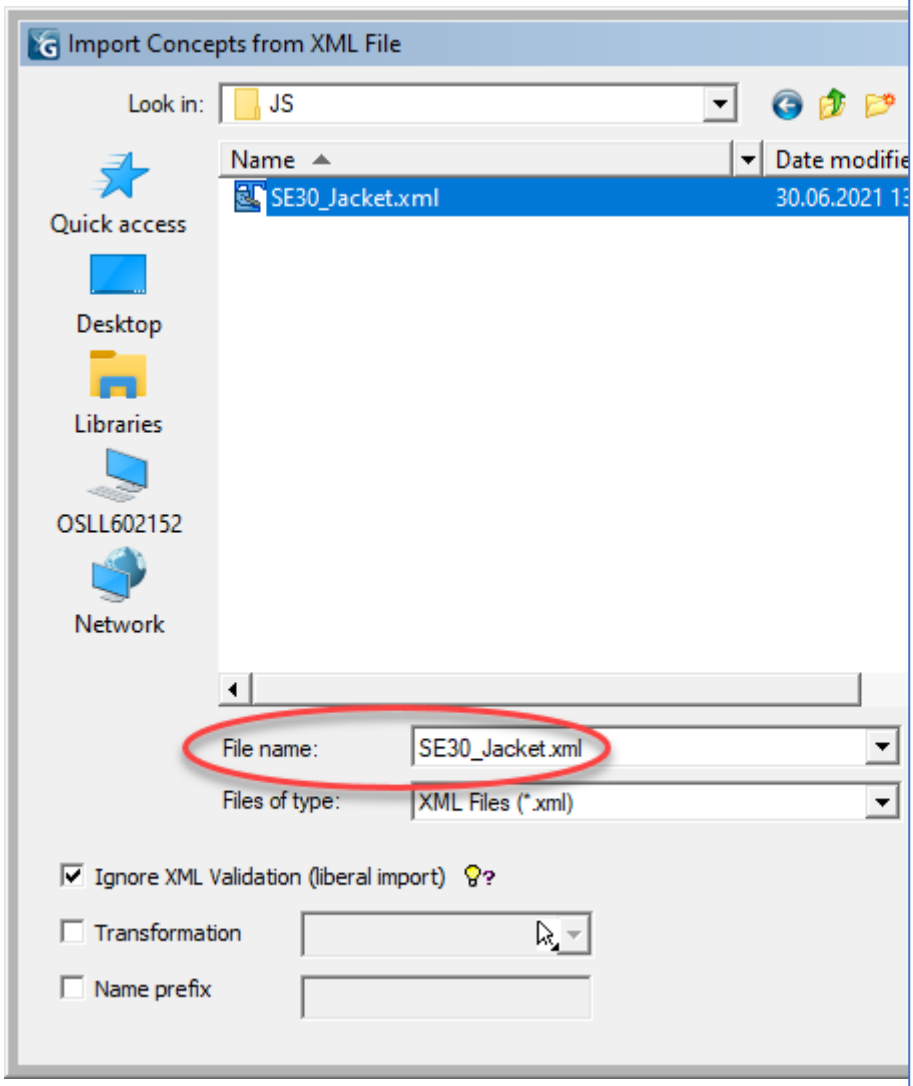
2 OPEN A NEW WORKSPACE AND IMPORT A JACKET MODEL

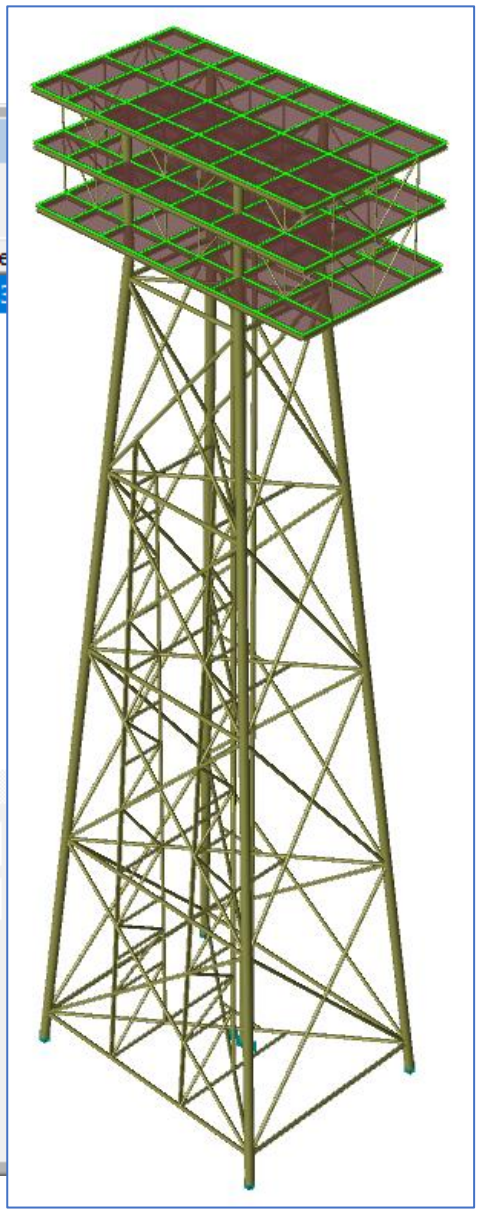


- Start GeniE and open a new workspace.
  - Give a workspace name, for example Topside_Wind_Loads.
  - Press the *Jacket mode* button to limit menus and buttons to those relevant for beam modelling.
  - Accept default units m and N and click OK.

- Use *File* | *Import* | *XML Concept Model* to import the file SE30_Jacket.xml. The file is found as part of the installation, typically at: C:\Program Files\DNV\GeniE VX.Y-ZZ\Help\Tutorials\TutorialsAdvancedModelling\A10_GeniE_Wind_Loads\JS

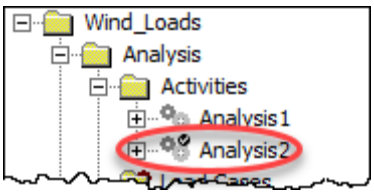
- The model shown to the lower right appears.



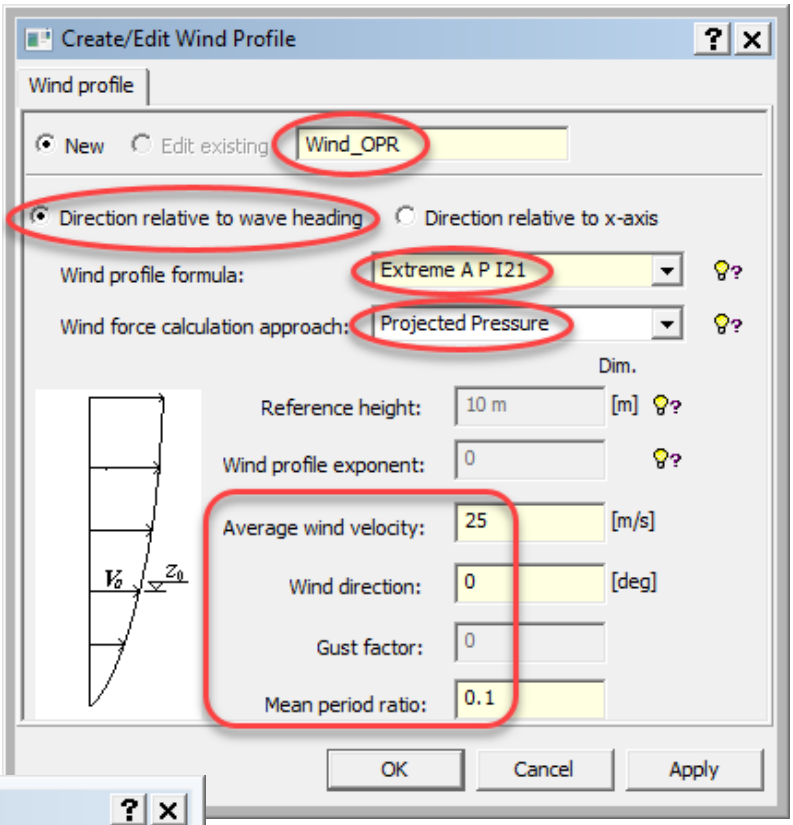


3 DEFINE WIND PROFILES FOR OPERATING AND STORM CONDITIONS

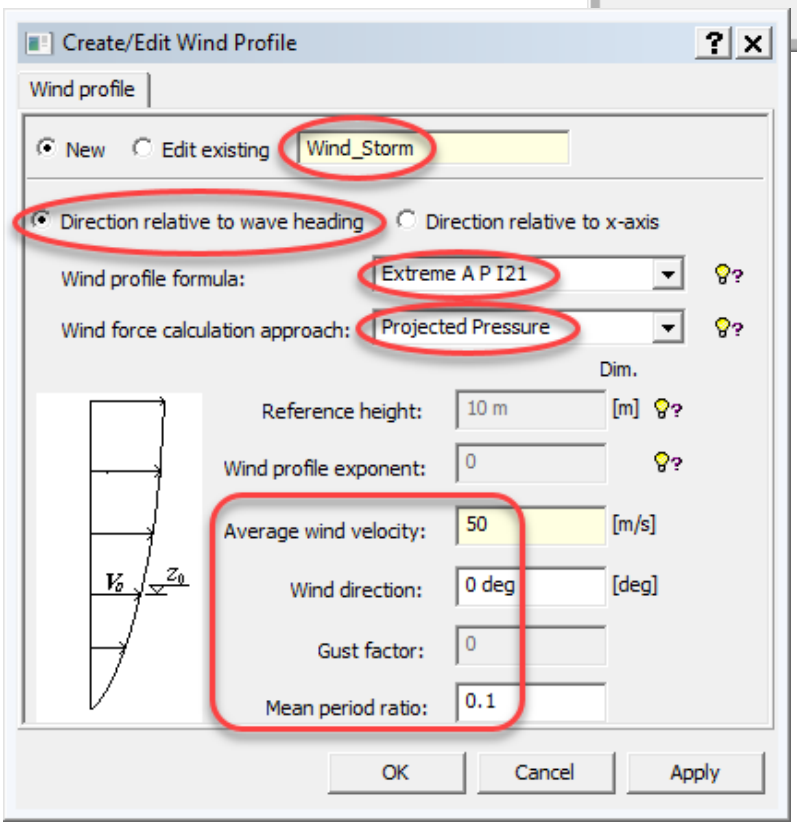
- Notice by the small check mark that Analysis2 is the currently active analysis.



- Right-click the *Environment* | *Air* folder in the browser and select *New Wind Profile* to create a new wind profile named Wind_OPR as shown to the right. The *Direction relative to wave heading* option combined with *Wind direction* 0 deg. means that the wind has the same direction as the wave. The average wind speed is 25 m/s.
  - There are several wind profile formulas available, *Extreme API 21* is based on API 21st edition.



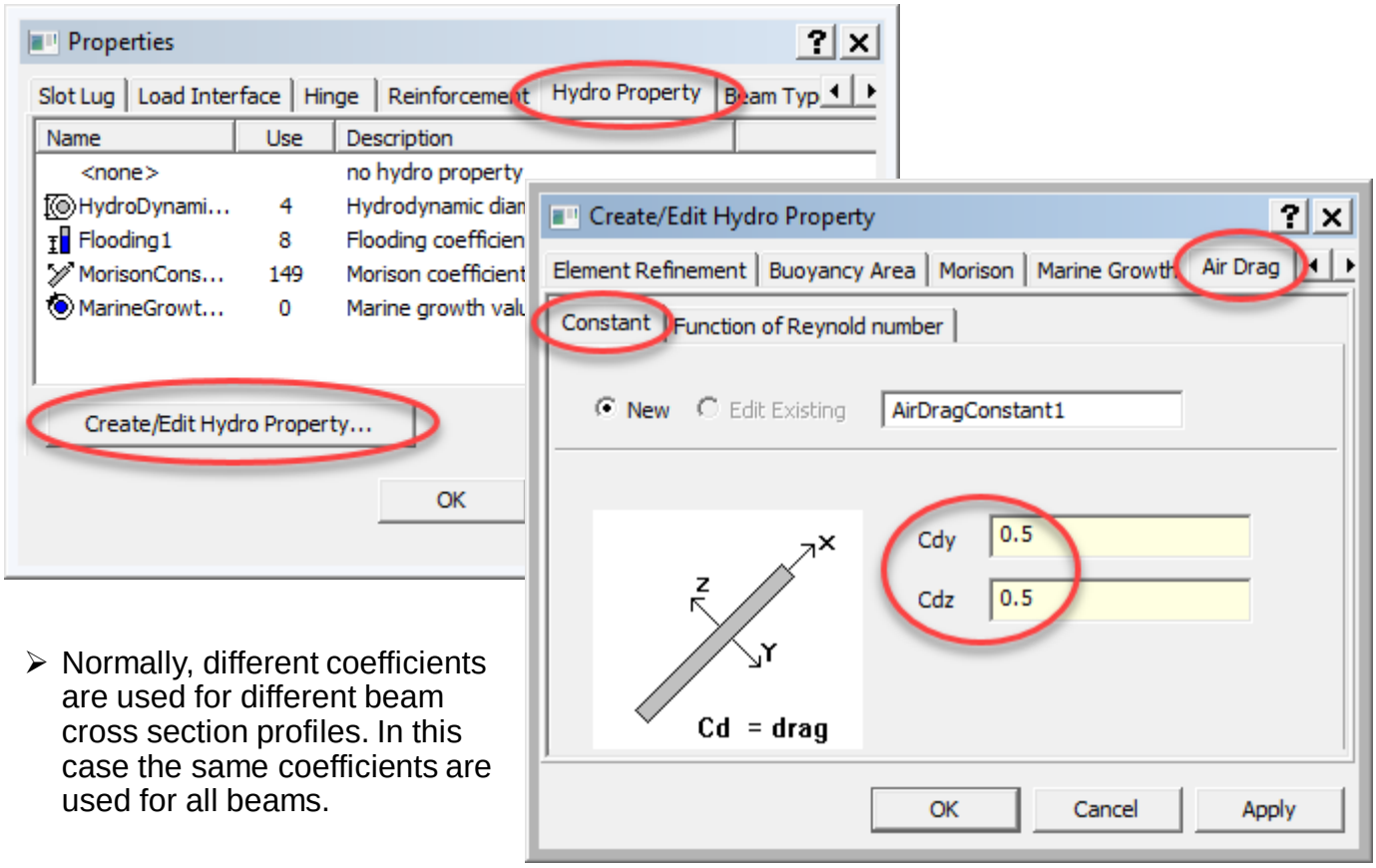
- Create a second wind profile named Wind_Storm with wind speed 50 m/s as shown below.



4 WIND LOADS ON BEAMS OF JACKET AND TOPSIDE COMPUTED BY WAJAC

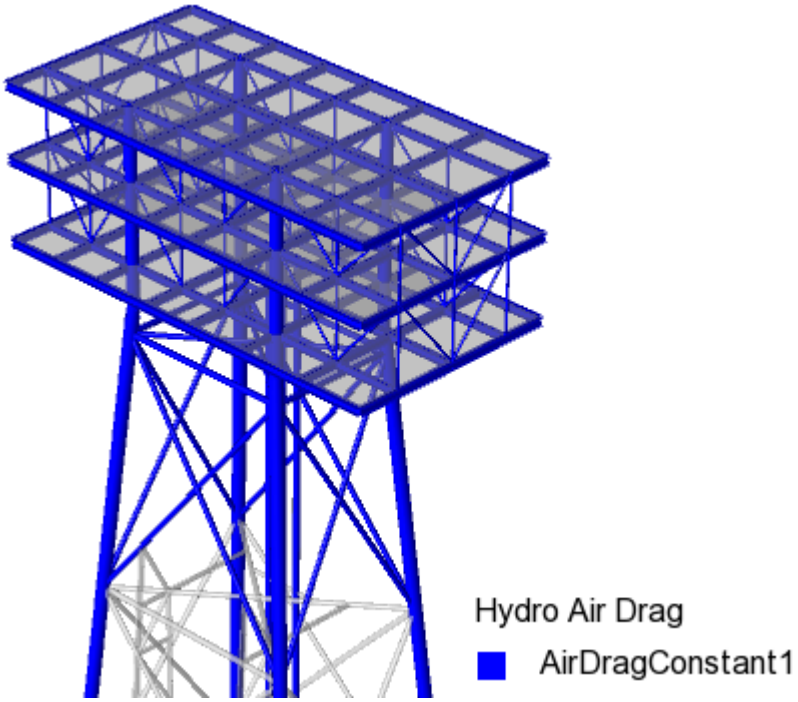
- Define air drag coefficients by *Edit* | *Properties*, go to the *Hydro Property* tab of the *Properties* dialog and click *Create/Edit Hydro Property*.

- In the *Create/Edit Hydro Property* dialog go to the *Air Drag* tab and define *Constant* coefficients in the two directions perpendicular to the beams, *Cdy* and *Cdz*.



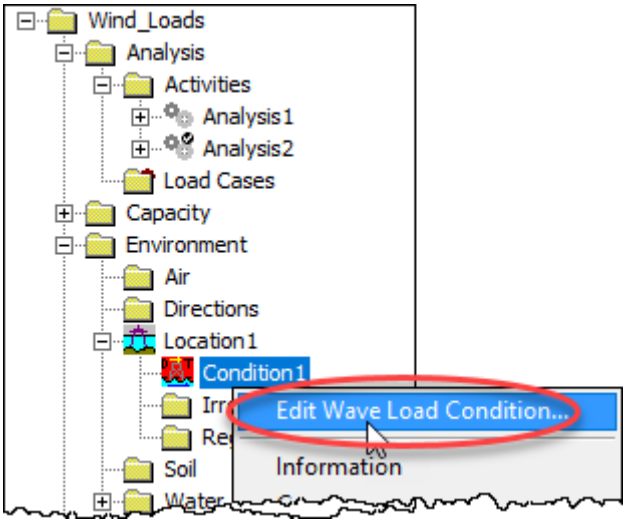
- Normally, different coefficients are used for different beam cross section profiles. In this case the same coefficients are used for all beams.

- Assign the air drag coefficients to the beams of the jacket and topside. Beams below the lowest wave trough need not be selected.
  - Air drag coefficients are found in the *Properties* | *Hydro* folder in the browser.



- Wind loads on beams are together with wave loads computed by Wajac. The imported model has already a wave load condition involving calculation of wave loads. Wind load profiles must be added to the wave loads.

  - Right-click the *Condition* folder, found under *Environment* | *Location1*, to *Edit Wave Load Condition* as shown to the right.



  - In the *Wind profile* column add Wind_Storm to the first three waves and Wind_OPR to the last three waves as shown below.

Edit Wave Load Condition

Deterministic Time

Name: Condition1

Wave components

Regular wave set: WaveSet1

Direction set: <Create new Direction

Frequency set: <Create new Frequer

Phase set: <Create new PhaseSe

Wave height set: <Create new WaveHe

Wave height function: <Create new prWave

Assign wave component properties

Current profile:

Wind profile:

Wave model:

Order:

Fill all

Fill selected

Fill equal components

Specify value:

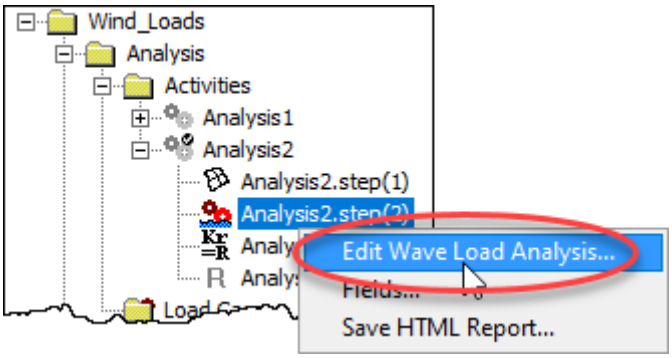
	Period	Height	Phase	Direction	Current profile	Wind profile	Wave model	Order
1	14 s	15 m	-2.384 deg	0 deg	CurrentProfile1	Wind_Storm	Stokes5	
2	14 s	15 m	-2.384 deg	37 deg	CurrentProfile1	Wind_Storm	Stokes5	
3	14 s	15 m	-2.384 deg	90.00000001 deg	CurrentProfile1	Wind_Storm	Stokes5	
4	12 s	12 m	-2.384 deg	0 deg	CurrentProfile2	Wind_OPR	Stokes5	
5	12 s	12 m	-2.384 deg	37 deg	CurrentProfile2	Wind_OPR	Stokes5	
6	12 s	12 m	-2.384 deg	90.00000001 deg	CurrentProfile2	Wind_OPR	Stokes5	
7							CalmSea	
8								

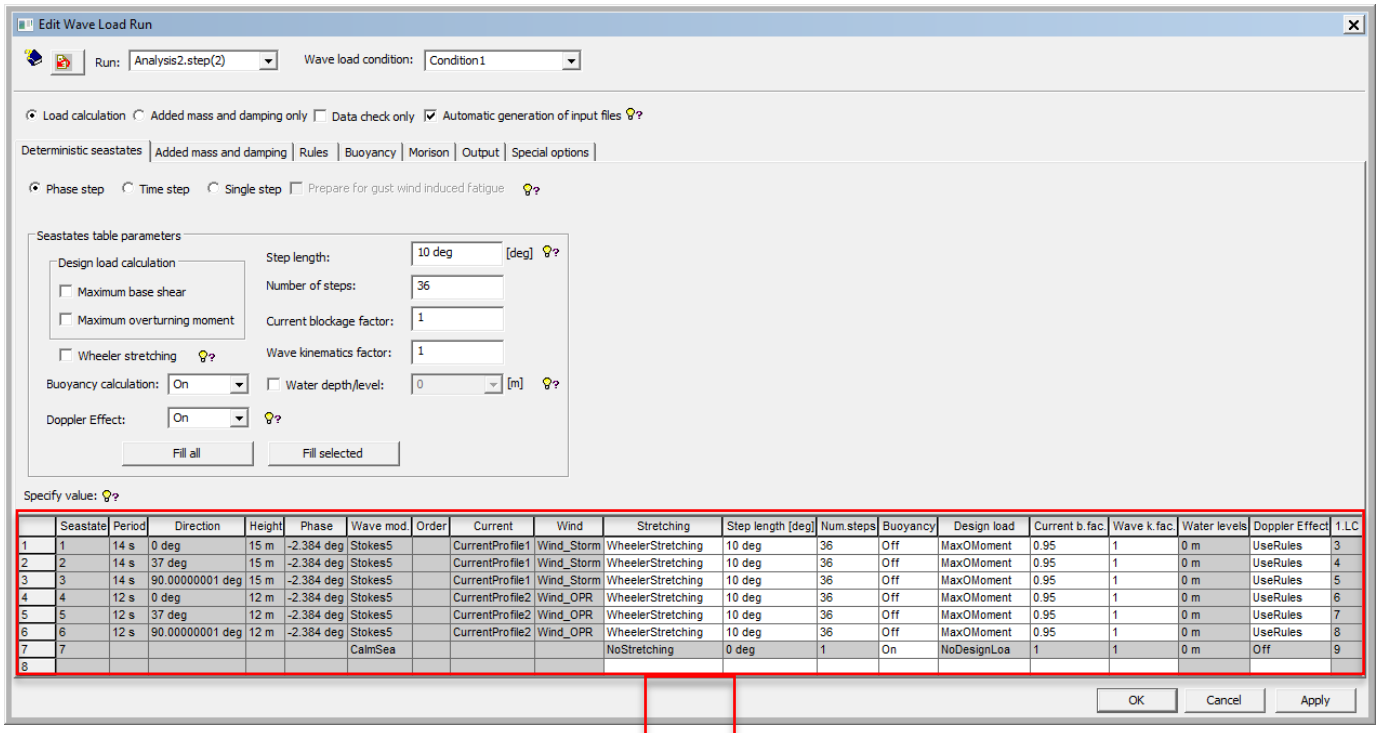
Number of rows with current selection: 7

Location: Location1

OK Cancel Apply

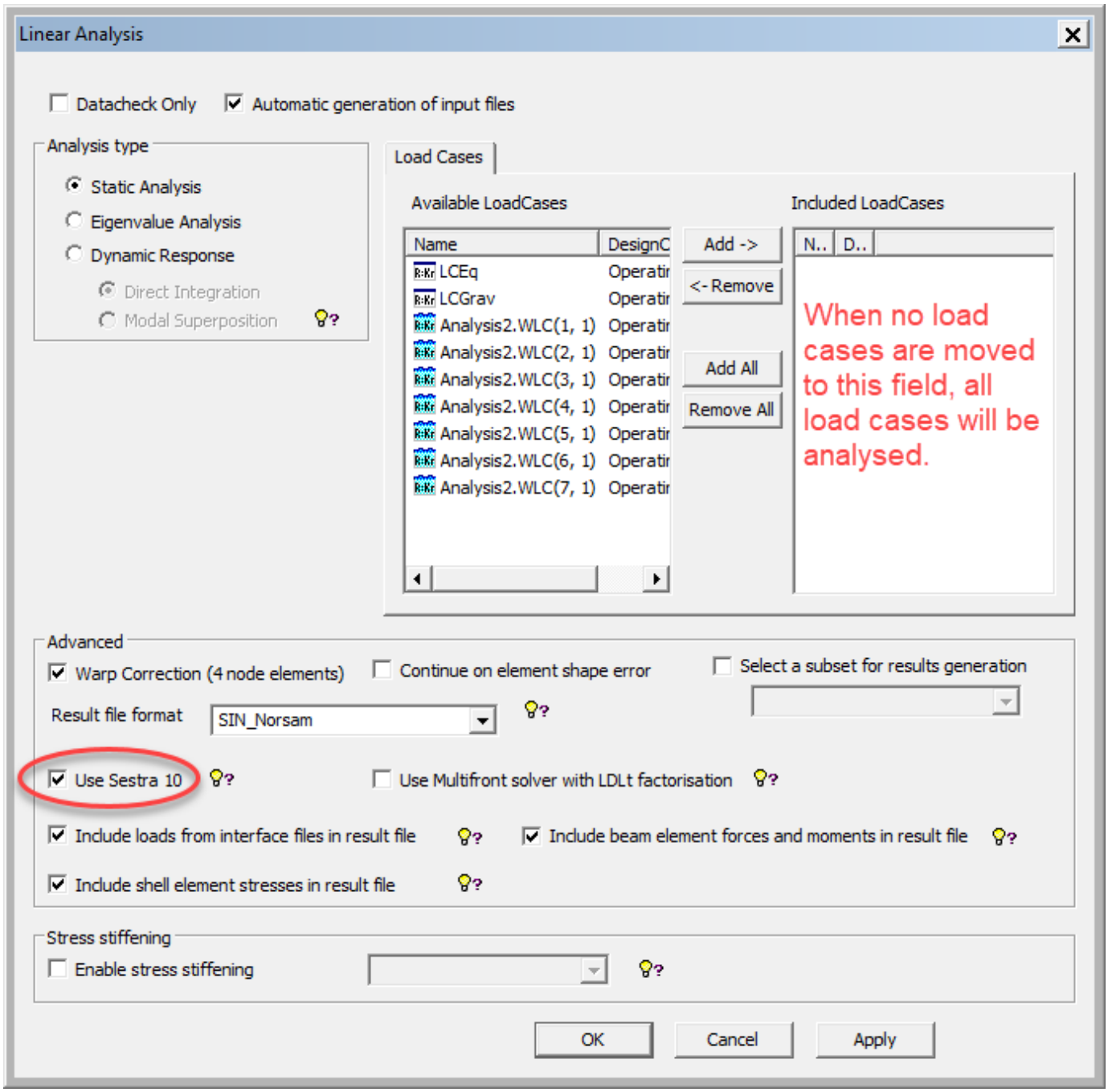
- Right-click the wave analysis activity of Analysis2 and click *Edit Wave Load Analysis* to open the *Edit Wave Load Run* dialog below. See that the waves will be stepped at 10 degree intervals 36 times and that the step saved for structural analysis in Sestra is the one giving maximum overturning moment (*MaxOMoment* in the *Design load* column). Do not make any changes in this dialog.





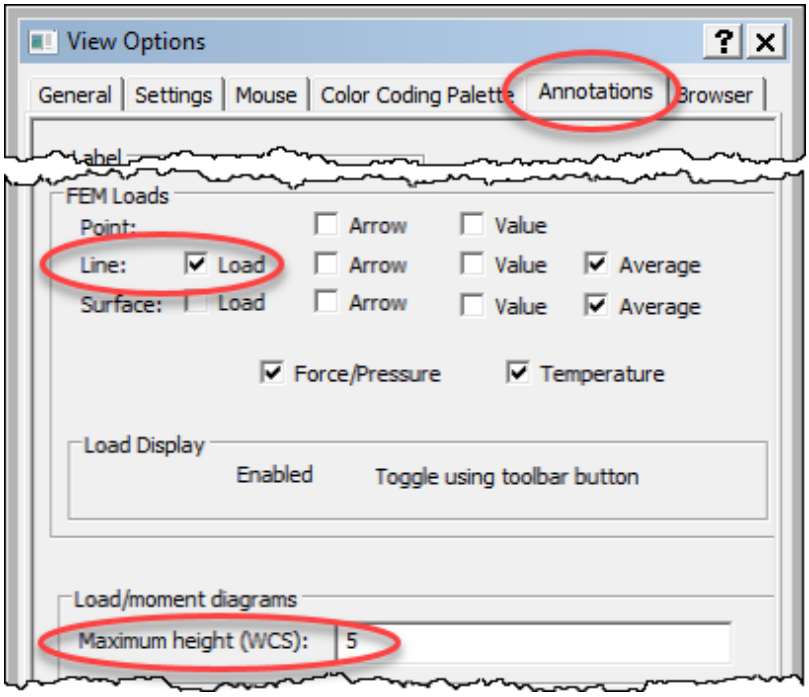
	Seastate	Period	Direction	Height	Phase	Wave mod.	Order	Current	Wind	Stretching	Step length [deg]	Num.steps	Buoyancy	Design load	Current b.fac.	Wave k.fac.	Water levels	Doppler Effect	1.LC
1	1	14 s	0 deg	15 m	-2.384 deg	Stokes5		CurrentProfile1	Wind_Storm	WheelerStretching	10 deg	36	Off	MaxOMoment	0.95	1	0 m	UseRules	3
2	2	14 s	37 deg	15 m	-2.384 deg	Stokes5		CurrentProfile1	Wind_Storm	WheelerStretching	10 deg	36	Off	MaxOMoment	0.95	1	0 m	UseRules	4
3	3	14 s	90.00000001 deg	15 m	-2.384 deg	Stokes5		CurrentProfile1	Wind_Storm	WheelerStretching	10 deg	36	Off	MaxOMoment	0.95	1	0 m	UseRules	5
4	4	12 s	0 deg	12 m	-2.384 deg	Stokes5		CurrentProfile2	Wind_OPR	WheelerStretching	10 deg	36	Off	MaxOMoment	0.95	1	0 m	UseRules	6
5	5	12 s	37 deg	12 m	-2.384 deg	Stokes5		CurrentProfile2	Wind_OPR	WheelerStretching	10 deg	36	Off	MaxOMoment	0.95	1	0 m	UseRules	7
6	6	12 s	90.00000001 deg	12 m	-2.384 deg	Stokes5		CurrentProfile2	Wind_OPR	WheelerStretching	10 deg	36	Off	MaxOMoment	0.95	1	0 m	UseRules	8
7	7					CalmSea				NoStretching	0 deg	1	On	NoDesignLoa	1	1	0 m	Off	9
8																			

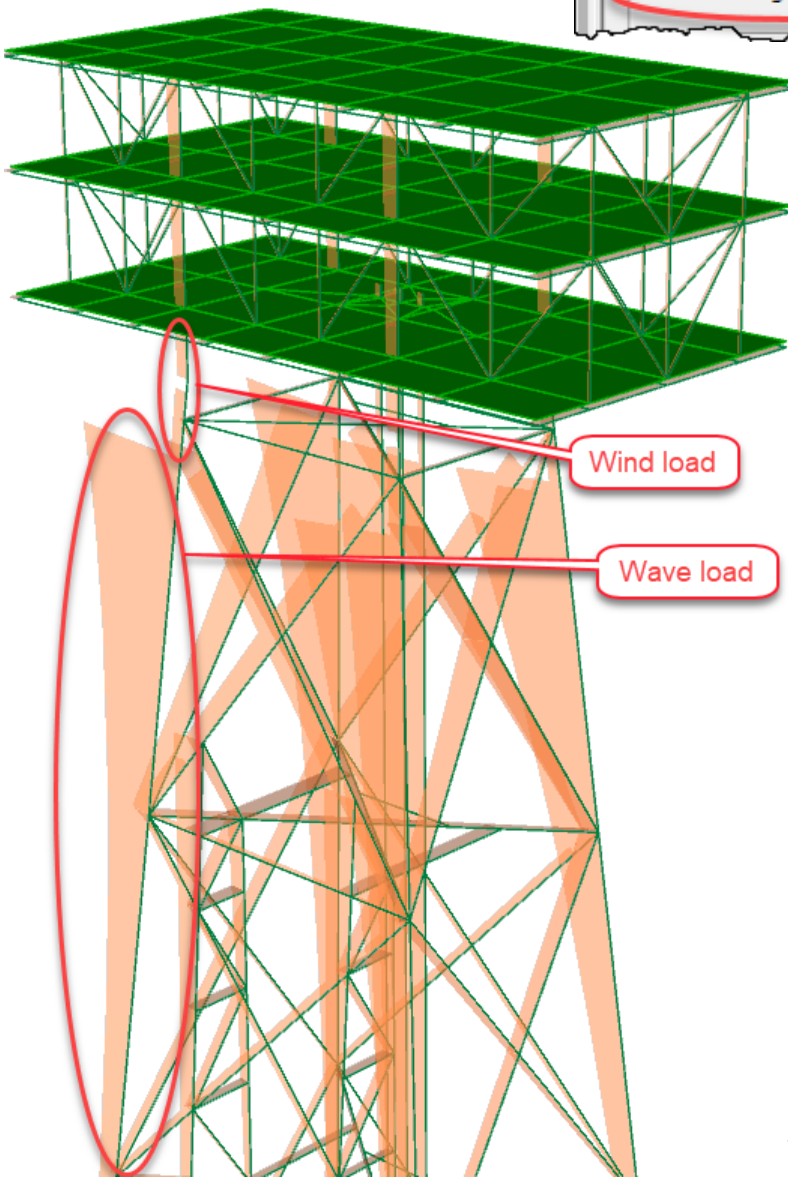
- With Analysis2 set as active use Alt+D to open the *Activity Monitor* and right-click the *Linear Structural Analysis, Static* activity (Sestra) to edit it.
  - In the *Linear Analysis* dialog, check *Use Sestra 10* (the imported workspace is a few years old and had the Sestra 8.8 version set as active).

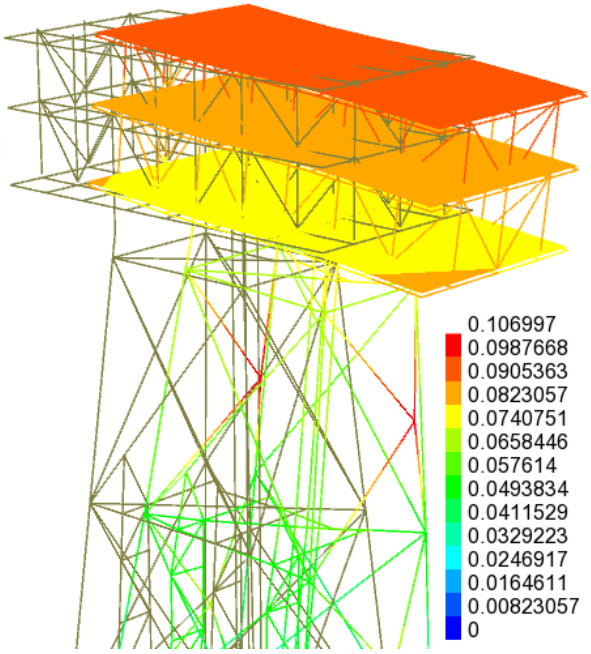


- Note that wind loads on plates and equipments are at this stage neglected.
- Run the analysis by clicking *Start* in the *Activity Monitor*.

- Switch to *Mesh - All* display configuration to display wave and wind loads computed by Wajac.
- Use *View* | *Options* | *Annotations* to open the *View Options* dialog. In the *Annotations* tab check *Line: Load* and scale up the display of the loads (*Maximum height*).
- The load case Analysis2.WLC(1, 1) is wave and wind from direction 0.



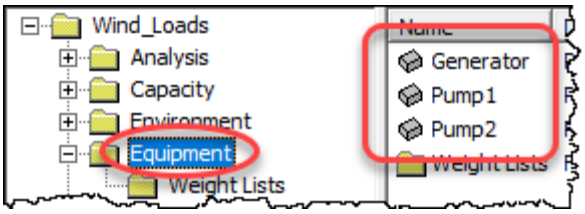




Deformed shape on top of undeformed for wave and wind from direction 0.

5 WIND LOADS ON EQUIPMENTS COMPUTED BY GENIE

- The imported model contains three equipments. These are found in the *Equipment* folder.

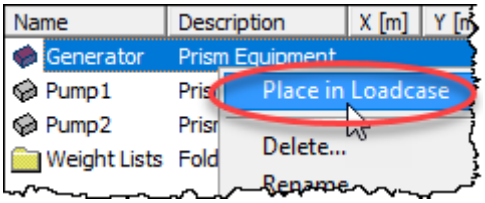


- Place the equipments in the load case LCEq. This load case will contain the self weight of the equipments.

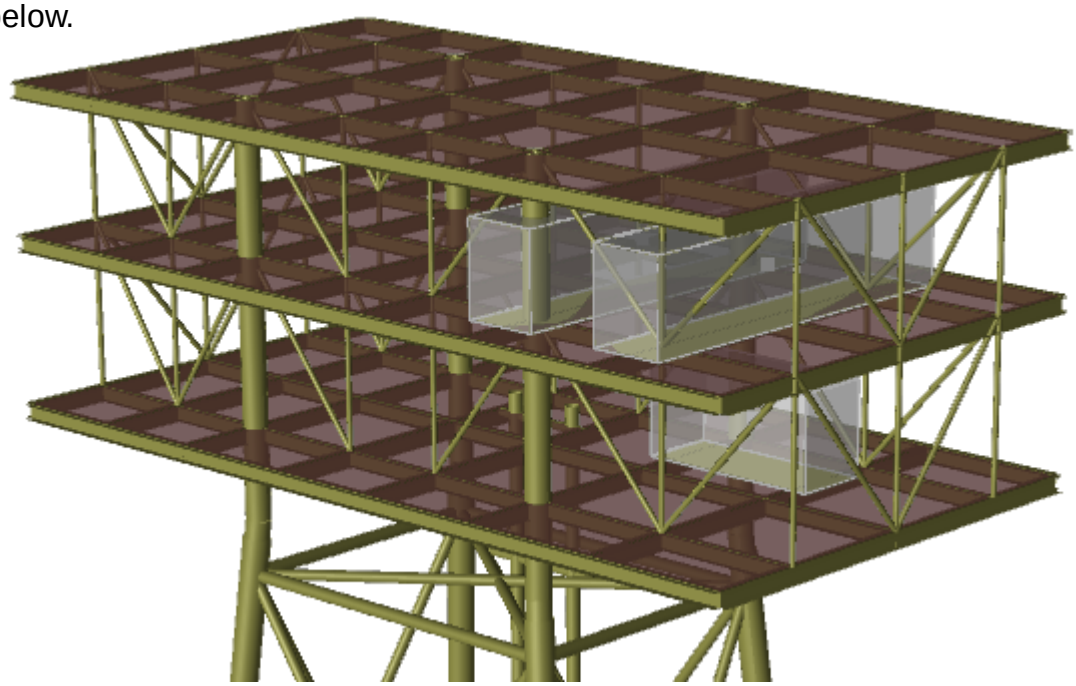
- The three equipments with their dimensions, masses and positions in the model are given in table below. Z-coordinate 19 is at the lower deck while Z-coordinate 28 is at the middle deck.

Equipment	Generator	Pump1	Pump2
Height (m)	5	5	5
Length (m)	10	4	4
Width (m)	4	20	20
Mass (tonne)	100	75	75
Position (m)	(14, 3.75, 19)	(18, 0, 28)	(10, 0, 28)

- Set LCEq as the current load case.
- Right-click an equipment and select *Place in Loadcase*.
- Click any position in the proper deck, lower deck in the case of Generator.



- Select and right-click the placed equipment, i.e. the one in the display area, to open the *Properties* dialog. Adjust the position of the equipment according to the table above.
- Repeat the process for the equipments Pump1 and Pump2. All three equipments are shown below.



- There are non-structural plates in the topside structure. But since these do not contribute with stiffness they have not been modelled. But they will be subjected to wind pressure and must be included somehow.
  - It is possible to create non-structural plates for this purpose.
  - The method chosen at this stage, however, is to create dummy equipments.
  - Use *Loads* | *Prismatic Equipment* to open the *Insert Equipment* dialog and define the two equipments tabulated below. The widths are irrelevant and just set to a low value. The Position row and Rotated row data are used to position the equipments later on.

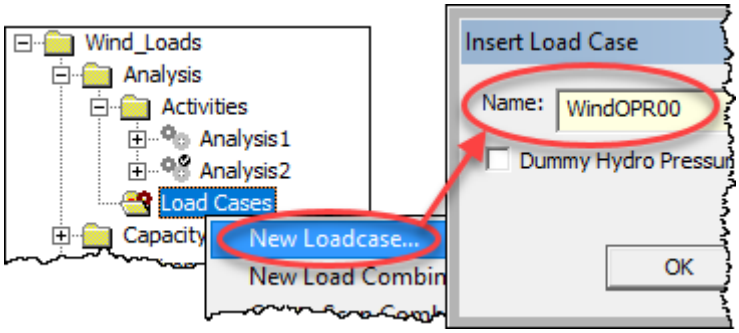
Equipment Name	Dummy1	Dummy2
Height (m)	6	5
Length (m)	48	25
Width (m)	1	1
Mass (tonne)	0	0
Position (m)	(2, −12.5, 36)	(−22, 0, 36)
Rotated	No	90 deg. abt. Z

- Note that equipments must be placed in *all* load cases to which they shall contribute.
  - Generator, Pump1 and Pump2 was, therefore, included in LCEq to contribute with self weight. And they must be included in the wind load cases since they are subjected to wind pressures.
  - Dummy1 and Dummy2, however, have zero mass, so they need not be included in load case LCEq. Moreover, Dummy1 will, as seen later, be positioned so that it will not be subjected to wind pressure in direction 0 deg. Correspondingly, Dummy2 will not be subjected to wind pressure in direction 90 deg.

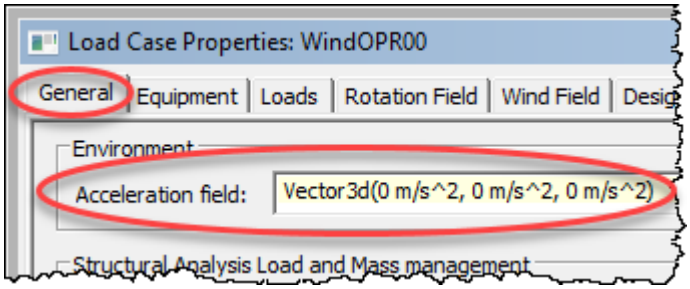
- The wind load cases tabulated below shall be created with wind directions in the same directions as the waves, and for operating and storm conditions.
  - The last column states which of the dummy equipments to include in the load cases.

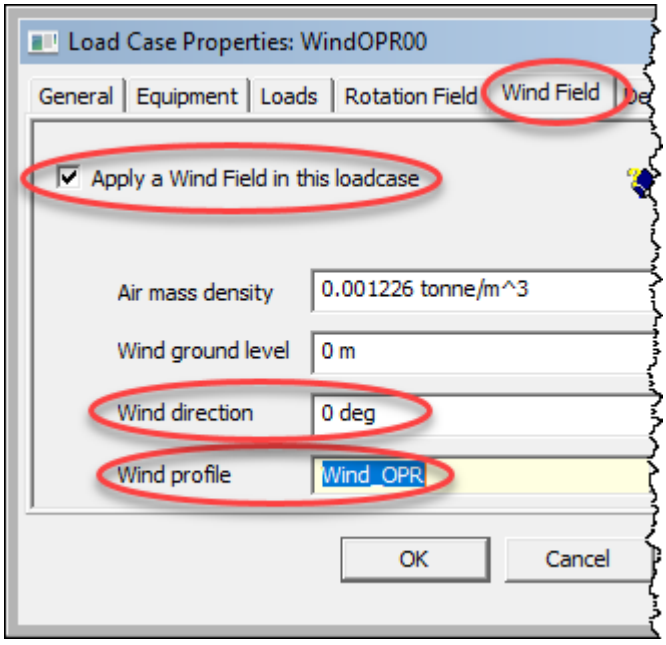
Load case	Wind profile	Wind direction	Design condition	Dummy equipment
WindOPR00	Wind_OPR	0	Operation	Dummy2
WindOPR37	Wind_OPR	37	Operation	Dummy1 & Dummy2
WindOPR90	Wind_OPR	90	Operation	Dummy1
WindSTM00	Wind_Storm	0	Storm	Dummy2
WindSTM37	Wind_Storm	37	Storm	Dummy1 & Dummy2
WindSTM90	Wind_Storm	90	Storm	Dummy1

- Create the load cases WindOPR00 by right-clicking the *Load Cases* folder. This puts the load case in that folder.
  - Load cases may also be stored in the folder of an analysis activity, e.g. Analysis2, but are then not available to other analysis activities.

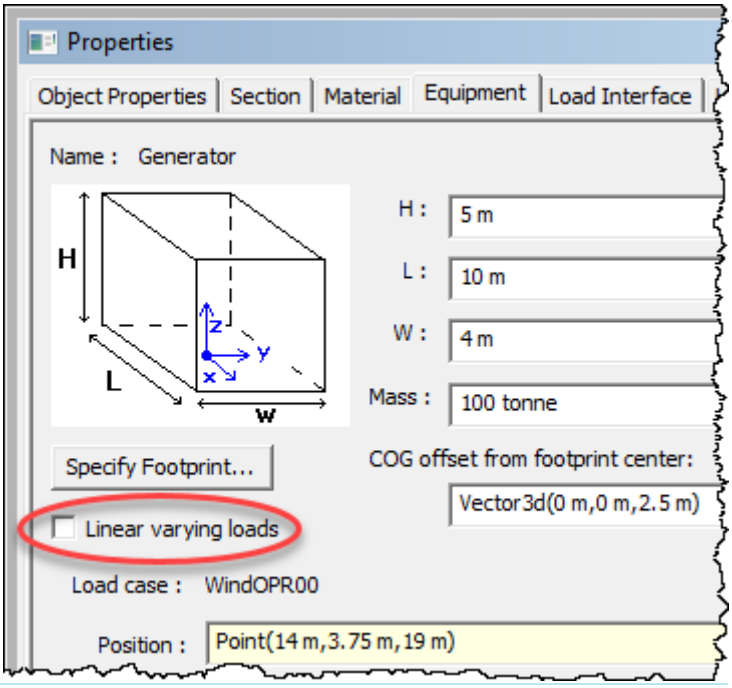


- Open the *Property* dialog for load case WindOPR00.
  - In the *General* tab below, set the *acceleration field* to (0,0,0) as we don't want the equipments to contribute with self weight.
  - In the *Wind Field* tab to the right, check *Apply a Wind Field in this loadcase*, let the wind direction be 0 degrees, and select the previously defined *Wind profile* Wind_OPR.



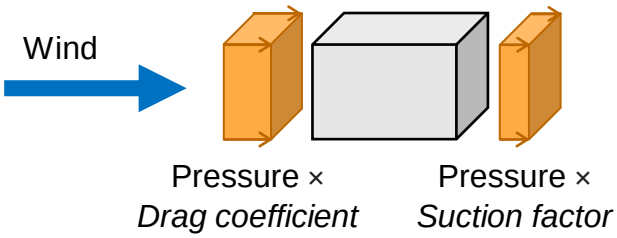


- Ensure that WindOPR00 is the current load case.
- Place the equipments Generator, Pump1 and Pump2 in the load case, positioning them as explained for LCEq.
  - While positioning them uncheck for each the option *Linear varying loads* as shown to the right. This involves that the position of the centres og gravity of the equipments are ignored in the calculation of line loads on the beams supporting the equipments. This has the effect that the line loads will only have horizontal components.

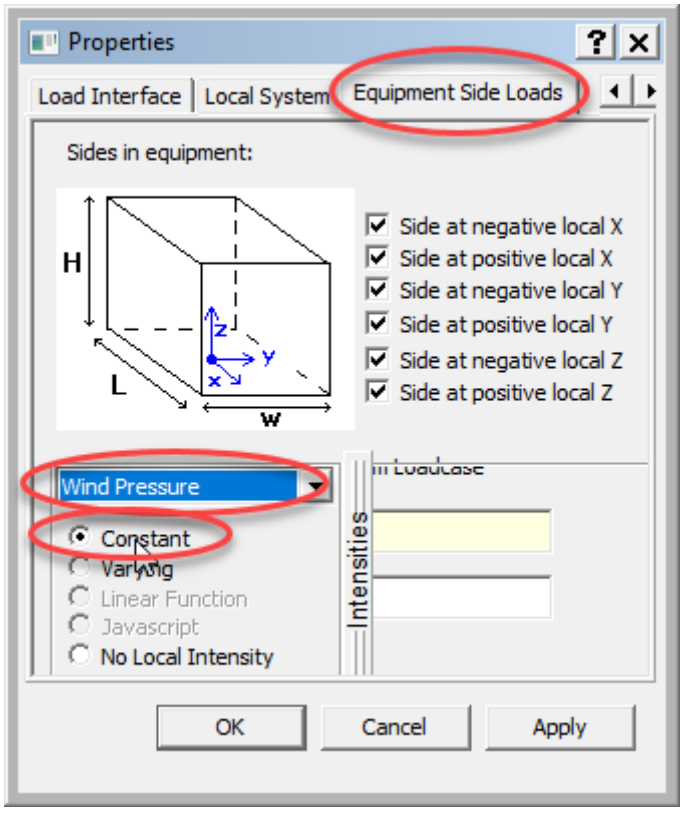


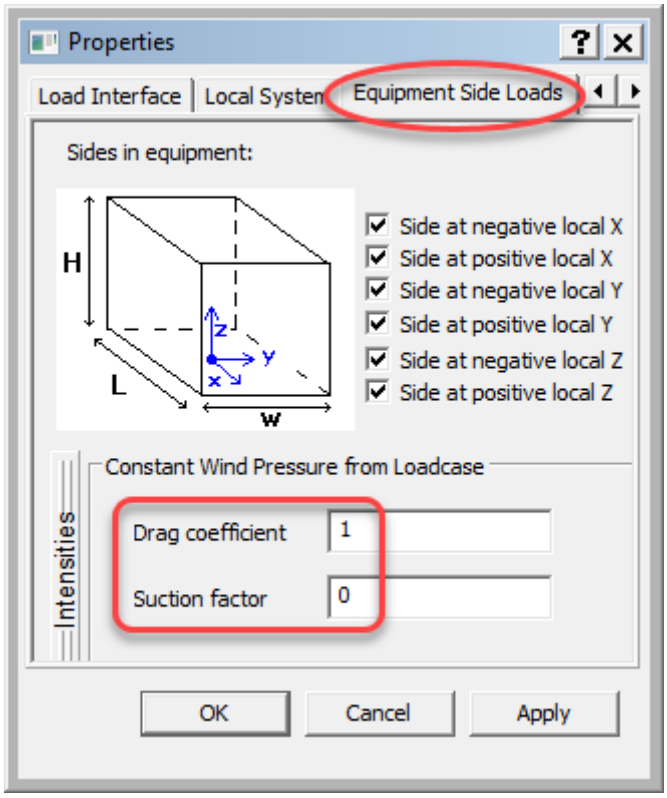
- We need to determine how the equipments shall be subjected to wind loads.

- Right-click the Generator equipment to open the *Properties* dialog and go to the *Equipment Side Loads* tab.
  - Rest the mouse on top of the *Intensities* 'curtain' to select *Wind Pressure* and *Constant* as shown in the *Properties* dialog to the left below.
  - Notice the *Drag coefficient* and *Suction factor* in the *Properties* dialog shown to the right below. The *Drag coefficient* is a factor (default value 1) for the wind pressure on the equipment side facing the wind. The *Suction factor* (default value 0) is a factor for the wind pressure on the equipment side facing away from the wind. Keep the default values.



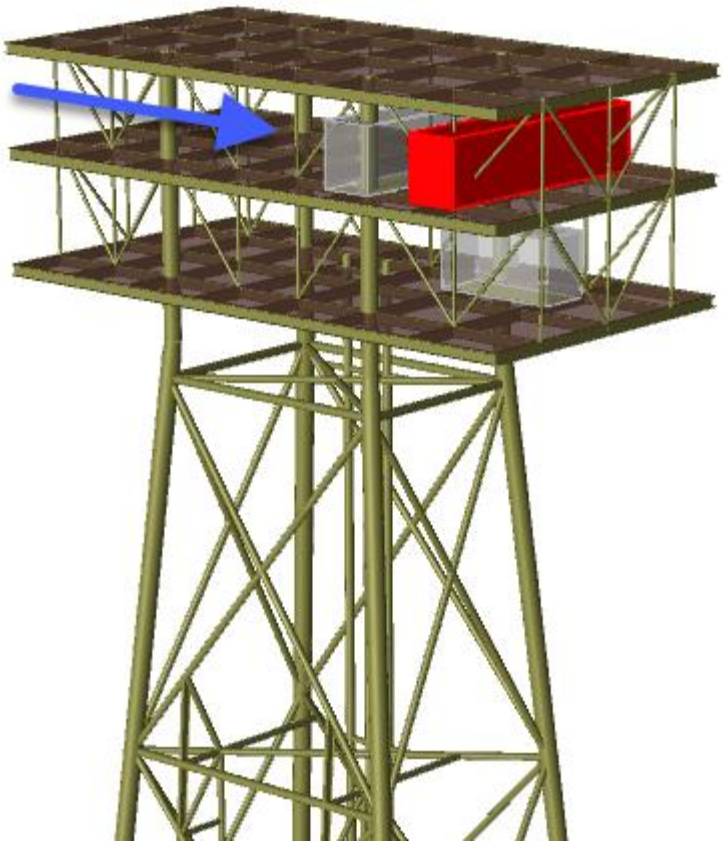
  - The dialog also allows deselecting sides from exposure to wind loads but since wind pressure will not be computed for sides that are parallel with the wind direction there is no need for deselecting sides in this case.





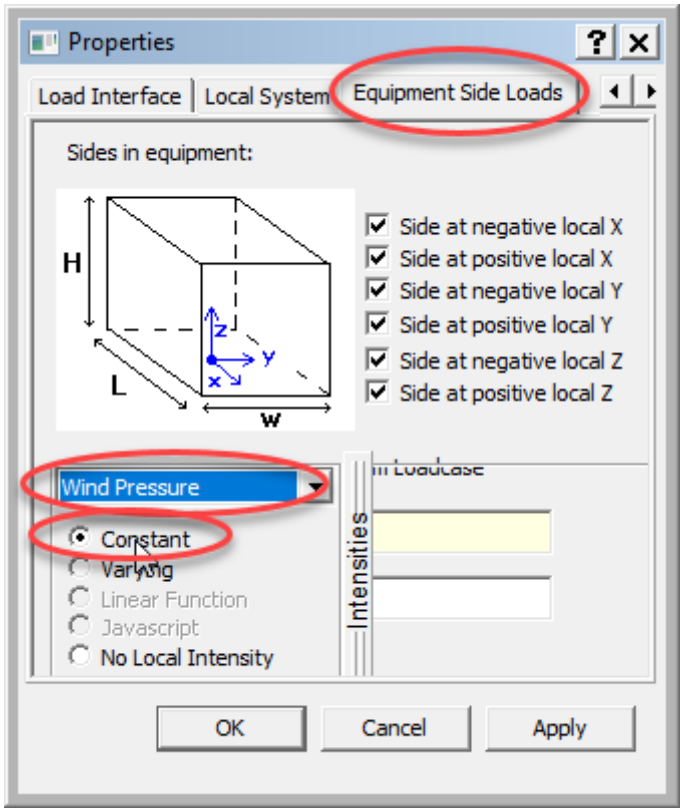
- Right-click the Pump2 equipment and repeat the process above.

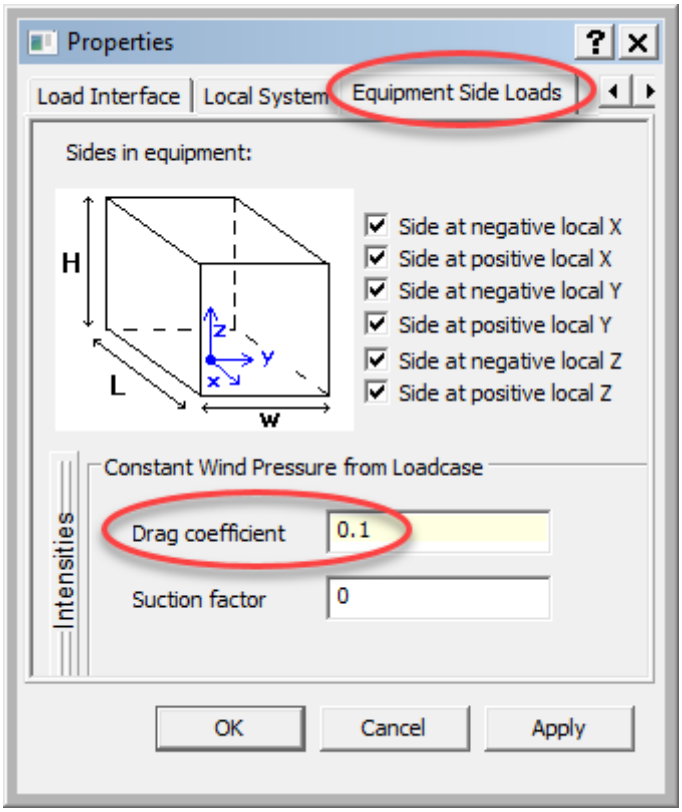
- With wind blowing in the X-direction (indicated by the blue arrow in the figure to the right) the equipment Pump1 (highlighted in red) will be shielded by the upwind equipment Pump2.



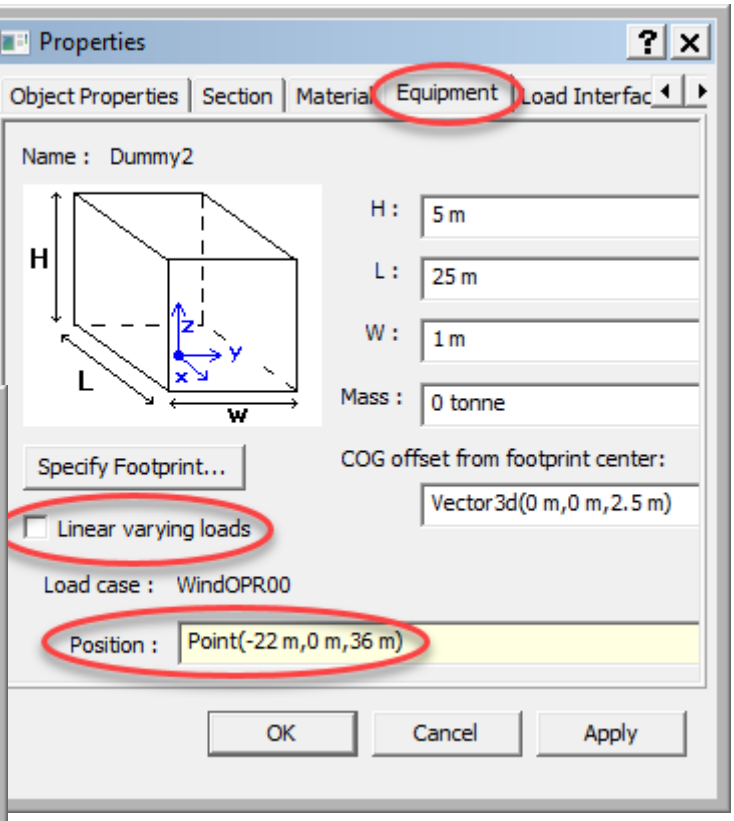
- This can be accounted for by reducing the drag coefficient.
  - Repeat the process of selecting *Wind Pressure* and *Constant* as shown below.
  - Then reduce the *Drag coefficient*. The value of 0.1 given in the figure to the right below is merely a suggestion and not based on any evaluation.

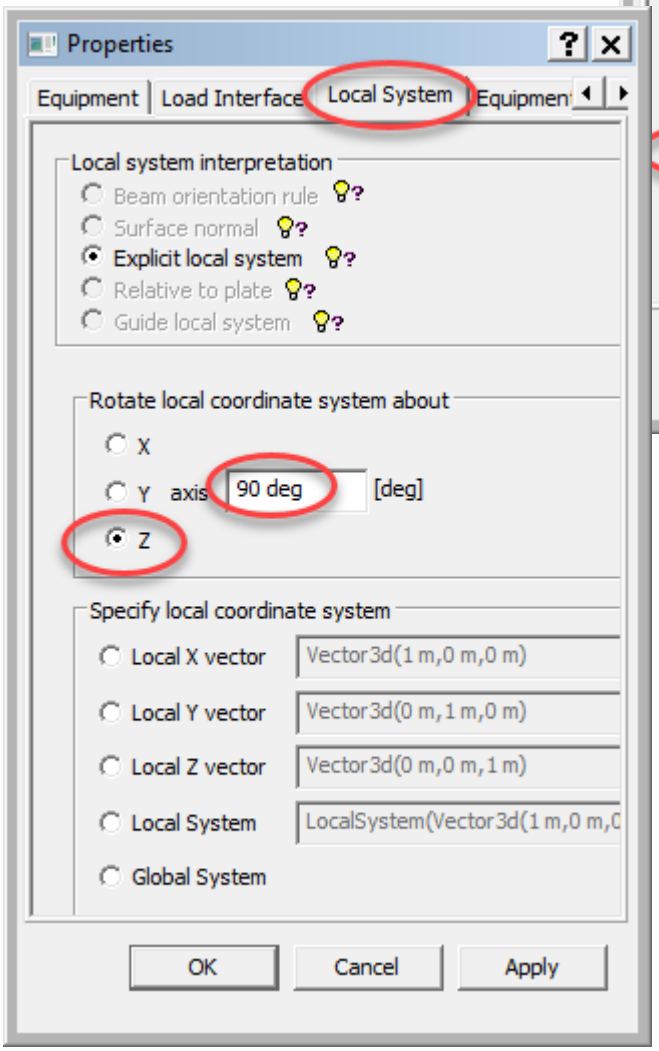
- A study may be done to determine proper values for both *Drag coefficient* and *Suction factor* for both the upwind and downwind equipments.





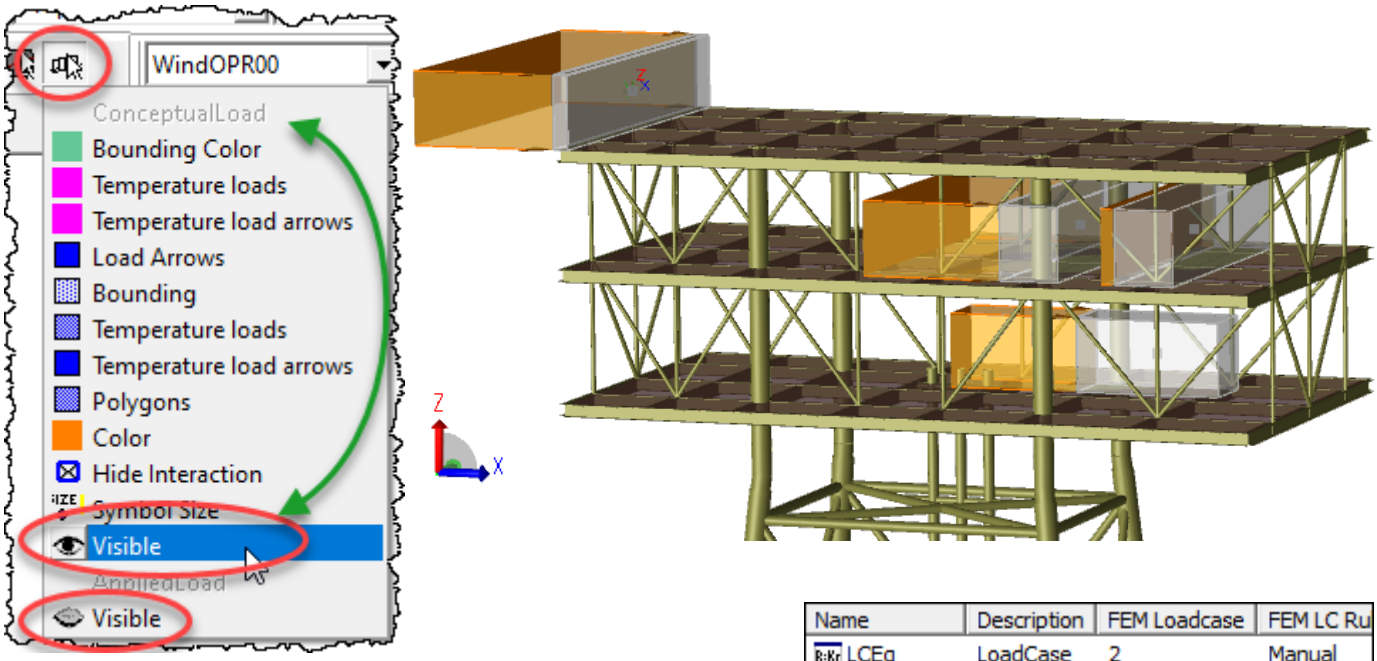
- Place the equipment Dummy2 in load case WindOPR00, positioning it as given in the table above introducing the dummy equipments.
  - Position it anywhere in the model, select it and give the proper position as shown to the right. Also uncheck *Linear varying loads* as done for the previous equipments. Click *OK* or *Apply*.
  - Reselect it to rotate it 90 deg. about the Z-axis as shown below.



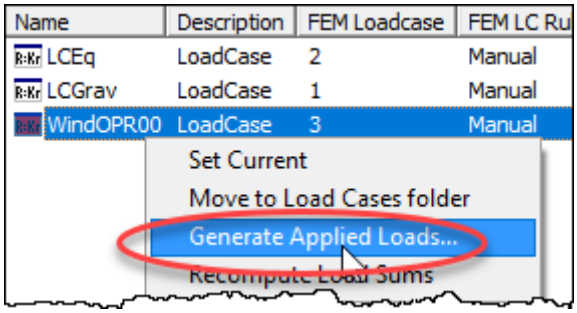


- Repeat the process of selecting *Wind Pressure* and *Constant* for the equipment Dummy2.
- Keep default values for the *Drag coefficient* and *Suction factor*.

- Right-click the *Load selection* button and open the eye symbol for the conceptual load as shown below to see the surface pressures on the equipments caused by the wind pressure.

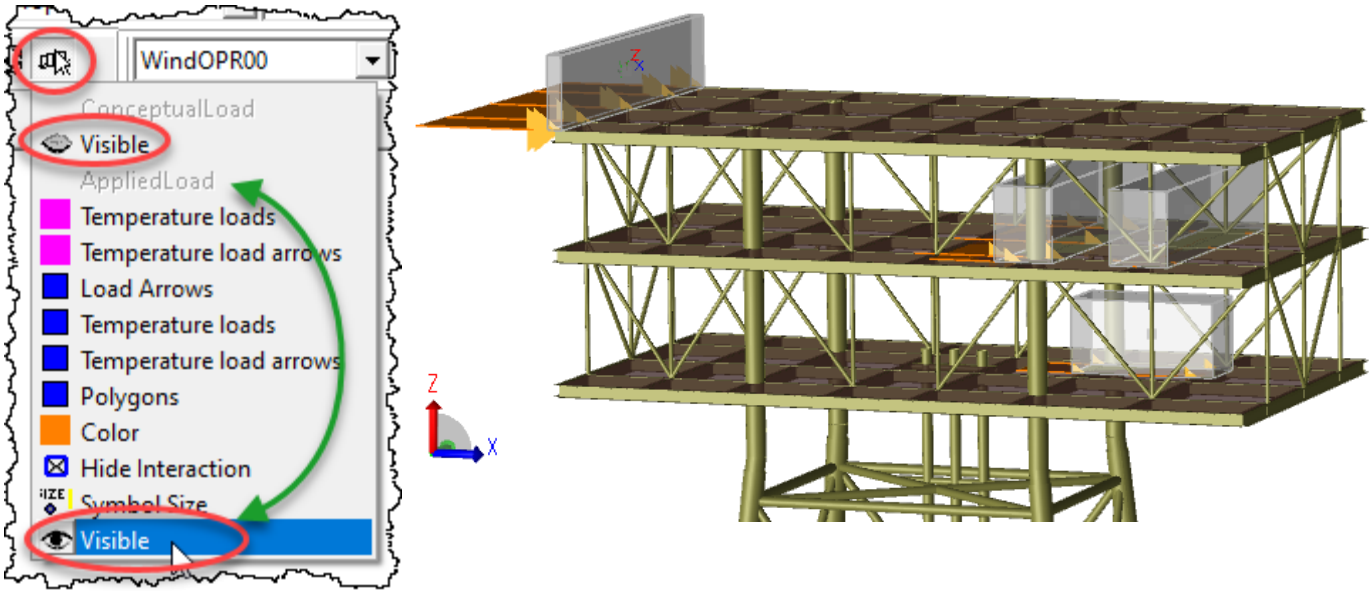


- To view the loads caused by the wind pressure applied as line loads on the beams supporting the equipments, right-click the load case in the *Load Cases* folder in the browser and select *Generate Applied Loads* as shown to the right.

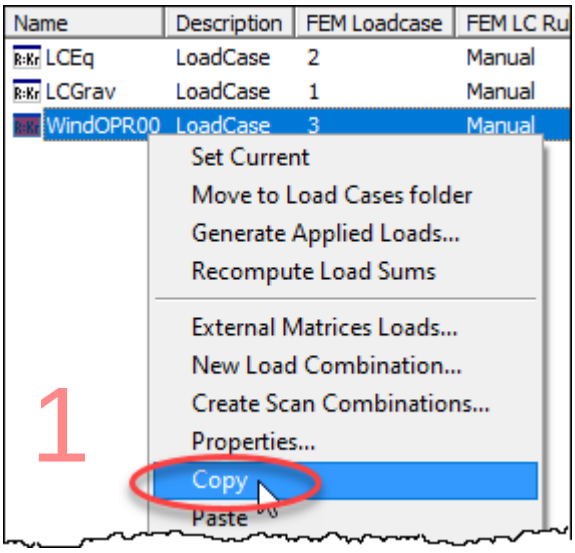


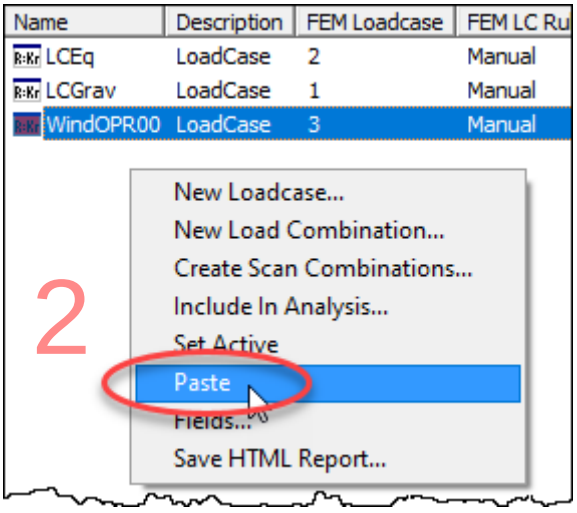
  - Note that this step above is only to display the applied beam line loads. These will anyhow be computed as part of an analysis.

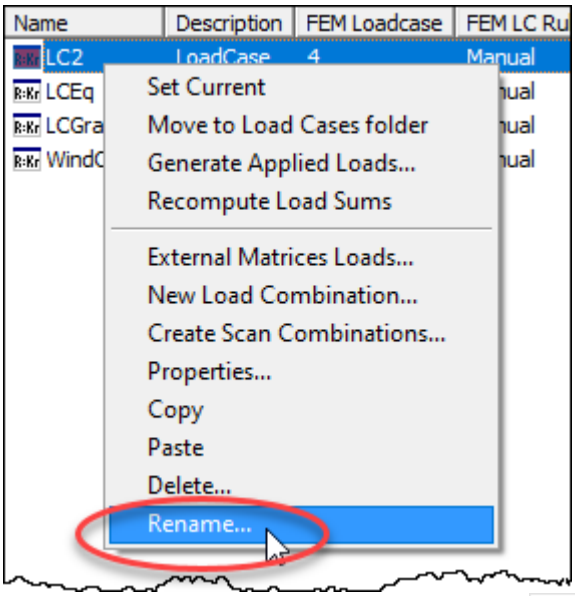
- Then shift to opening the eye symbol for the applied load to display the line loads applied to the beams supporting the equipments.

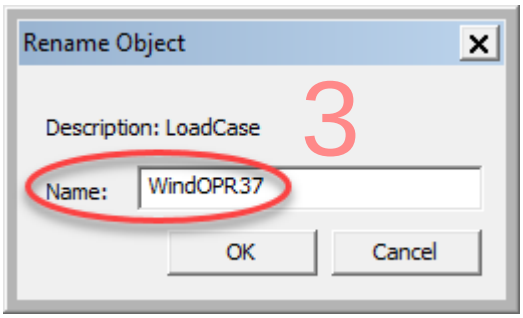


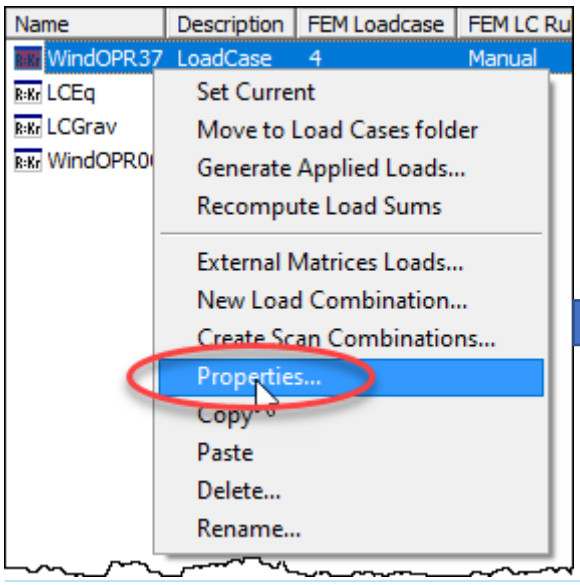
➢ Create the load case WindOPR37 by copying WindOPR00. Equipments with properties will be copied. The process is copy, paste, rename and change wind direction to 37 deg.

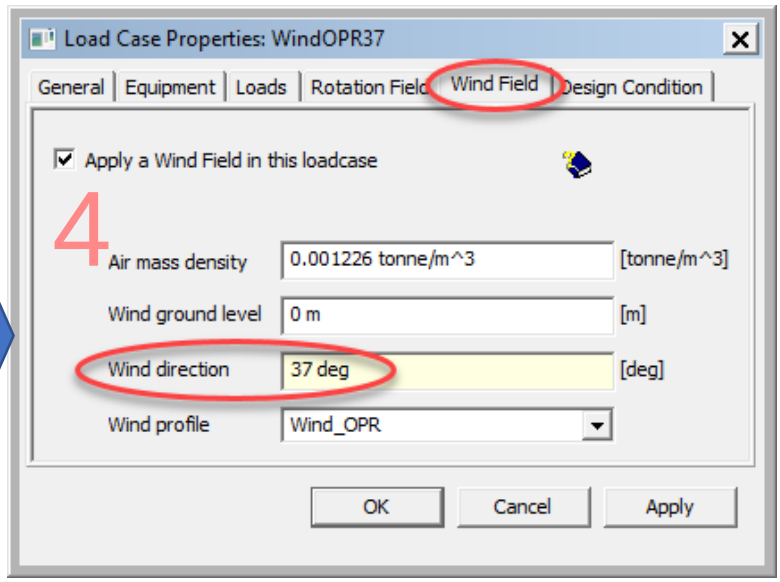




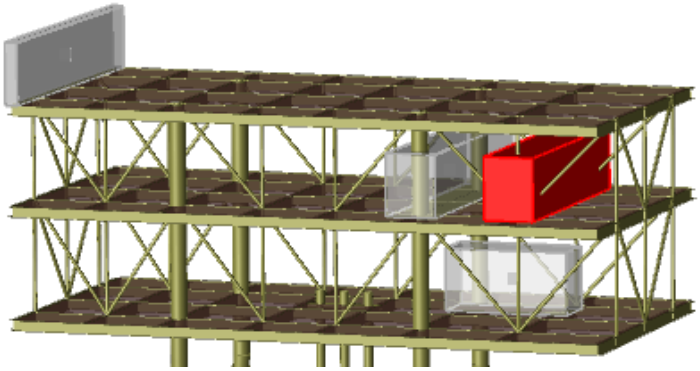


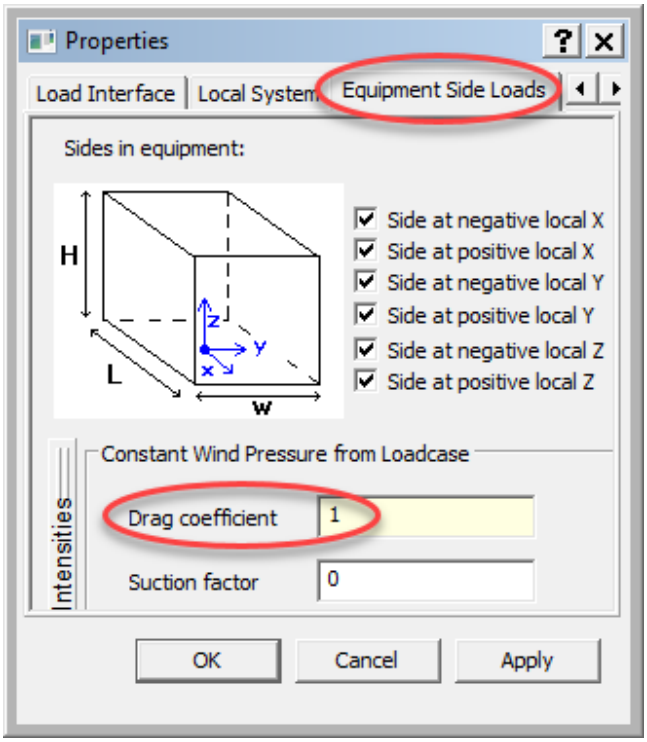




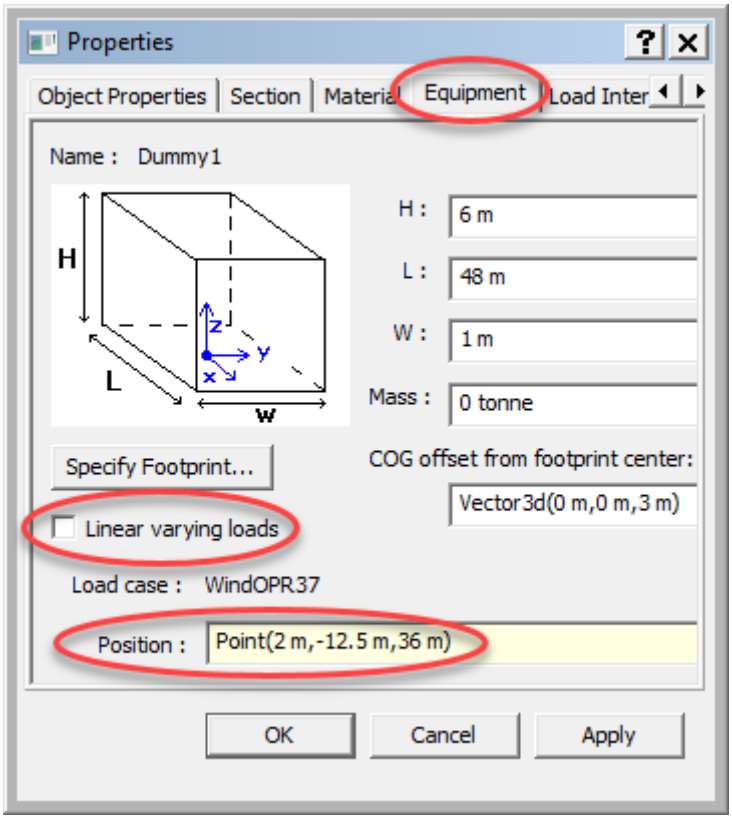


- Open the *Properties* dialog for the equipment Pump1 (highlighted below) and change the *Drag coefficient* back to 1 since Pump1 is no longer (completely) shielded by Pump2 when wind direction is 37 deg. This is a conservative choice.
  - Again, a study may be done to determine proper values for both *Drag coefficient* and *Suction factor* for both the upwind and downwind equipments.



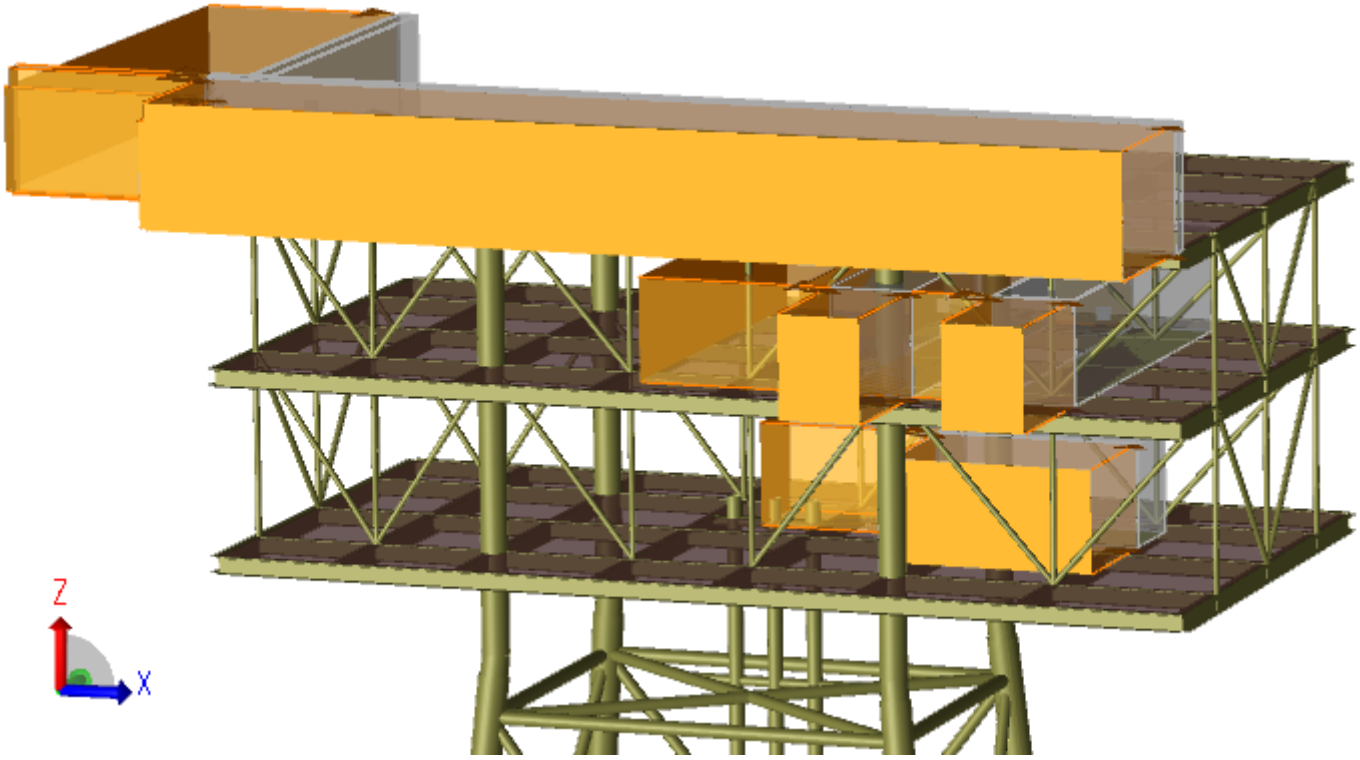


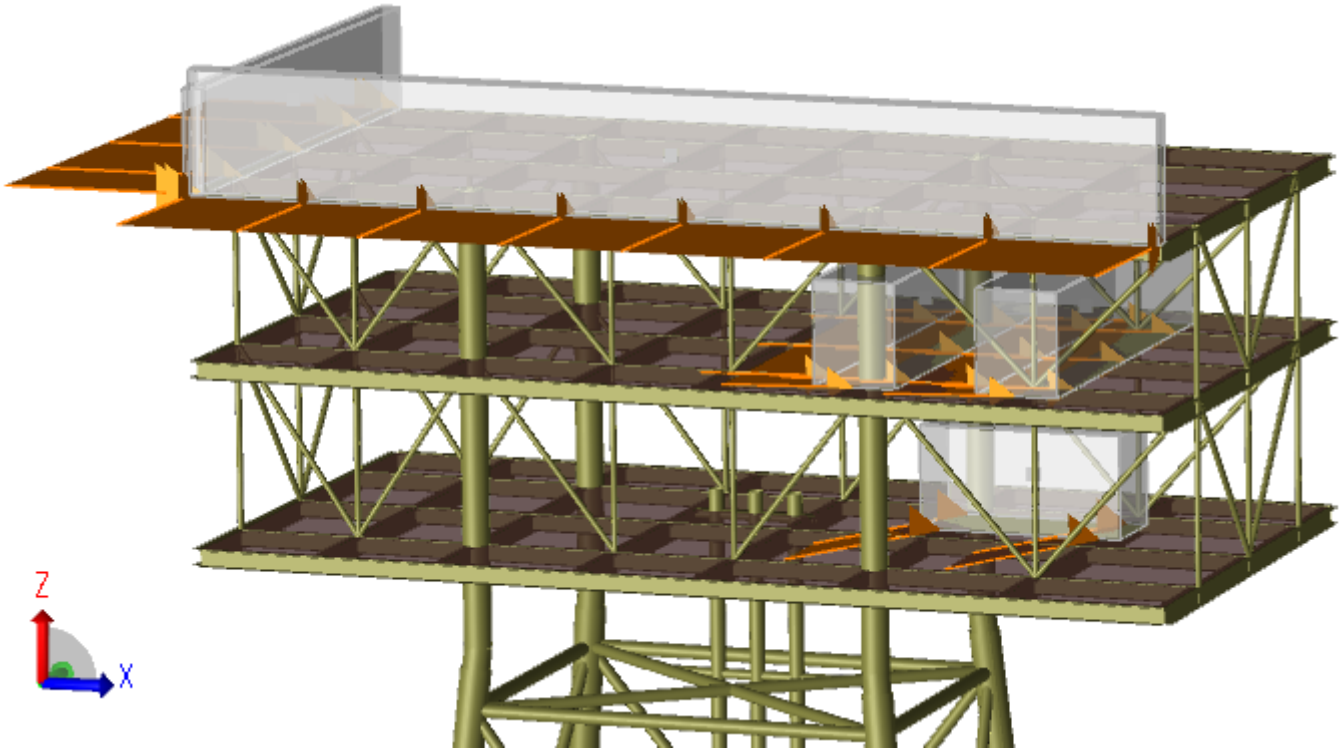
- Place the equipment Dummy1 in the load case. (Both equipments Dummy1 and Dummy2 contribute to the load case WindOPR37.)
  - Positioning Dummy1 as given in the table above introducing the dummy equipments.
  - Also uncheck *Linear varying loads*. Click *OK* or *Apply*.
- Repeat the process of selecting *Wind Pressure* and *Constant* for the equipment Dummy1.
- Keep default values for the *Drag coefficient* and *Suction factor*.



- View the applied beam line loads caused by the wind by right-clicking the load case in the *Load Cases* folder in the browser and selecting *Generate Applied Loads*.

- The conceptual loads and applied beam line loads are displayed below.

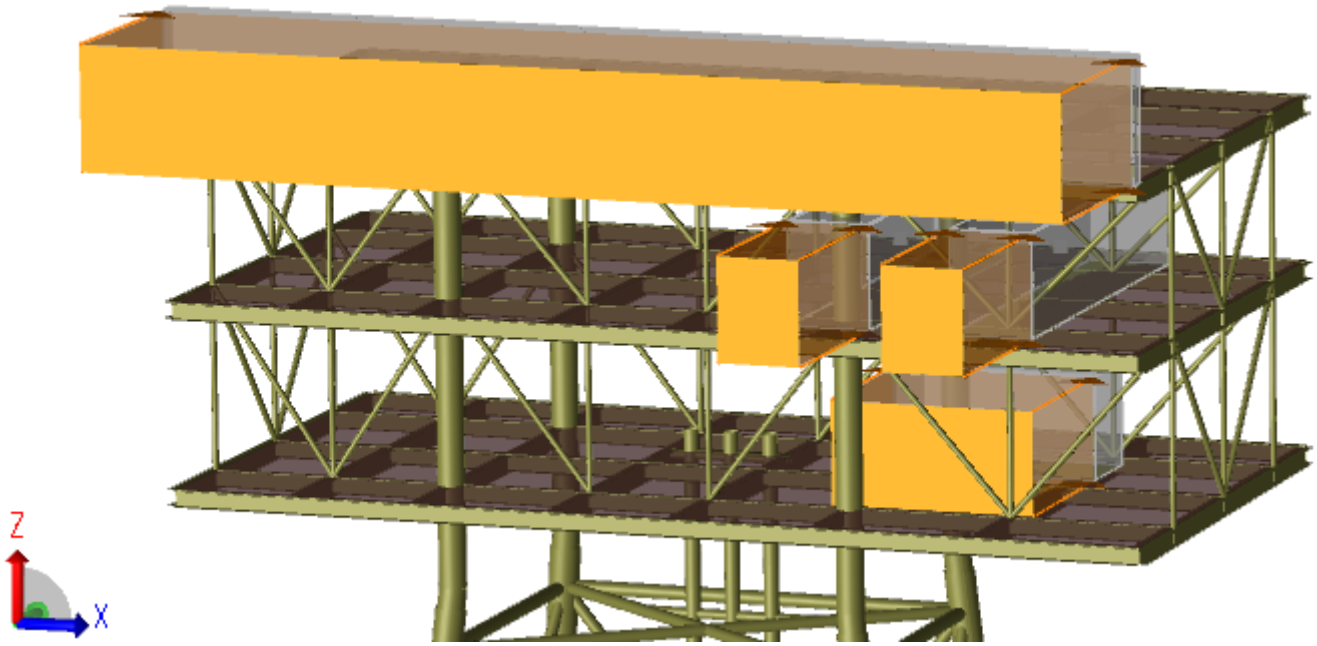


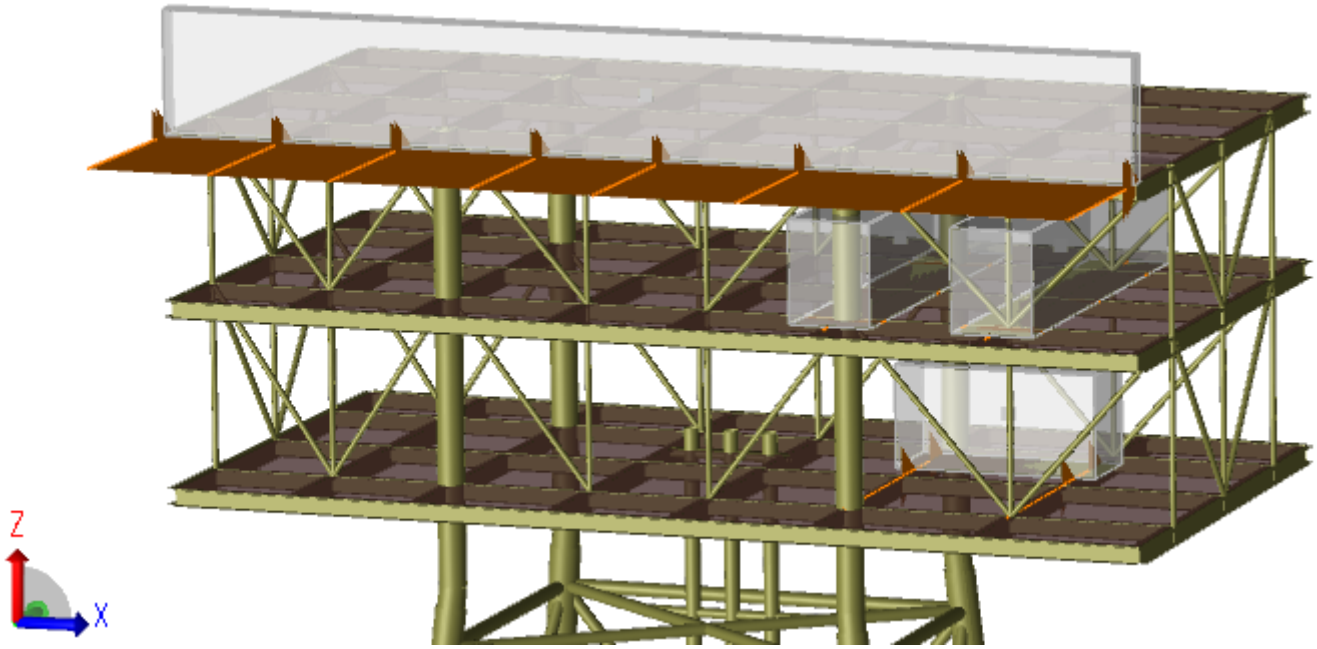


- Create the load case WindOPR90 by copying WindOPR37 and renaming it. Change wind direction to 90 deg.

- Delete the equipment Dummy2 which is parallel with the wind direction by selecting it in the display and pressing the Delete key.

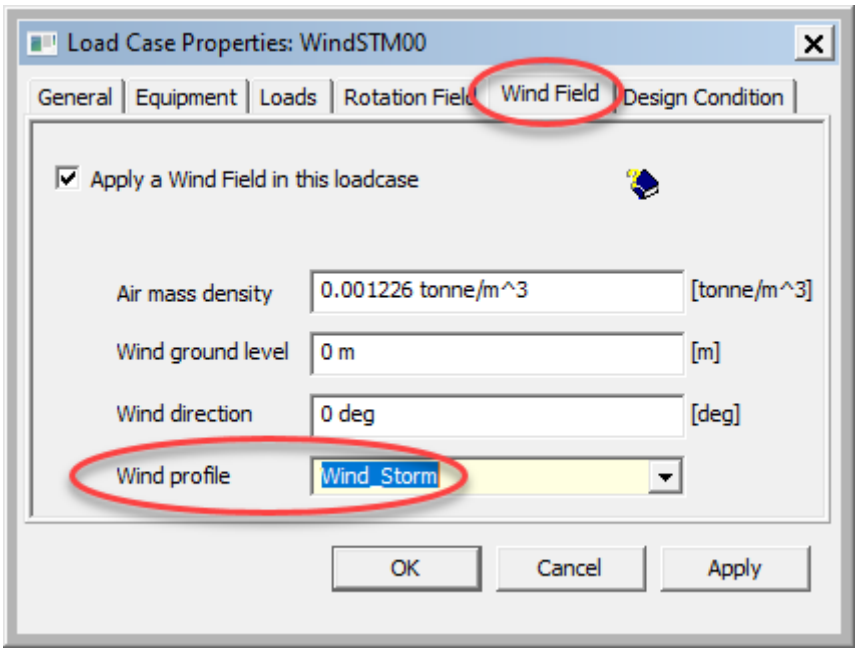
- The conceptual and applied wind loads are displayed below.





- Copy and paste wind load cases WindOPR00, WindOPR37 and WindOPR90, and rename them to WindSTM00, WindSTM37 and WindSTM90, respectively.

- Open the *Properties* dialog for each and change *Wind profile* Wind_Storm for all, shown below for WindSTM00.



- Verify the conceptual and applied wind loads for each.

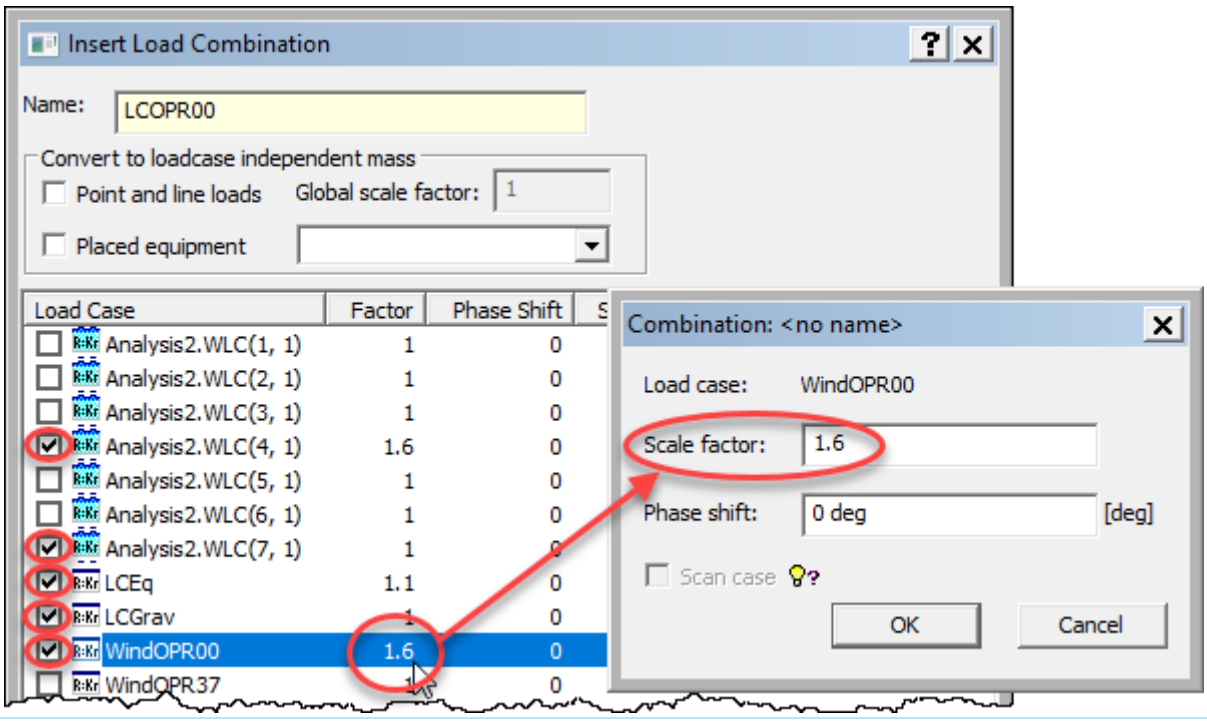
6 CREATE LOAD COMBINATIONS

- The equipment wind loads computed by GeniE must be combined with corresponding wave and wind loads on beams computed by Wajac. Moreover, load cases like structure self weight, equipment weight and buoyancy must be added to the combinations. Load factors are used for the various load cases as shown in the table below.

Load comb.	LCGrav	LCEq	WindOPR00	WindOPR37	WindOPR90	WindSTM00	WindSTM37	WindSTM90	Analysis2. WLC (1,1)	Analysis2. WLC (2,1)	Analysis2. WLC (3,1)	Analysis2. WLC (4,1)	Analysis2. WLC (5,1)	Analysis2. WLC (6,1)	Analysis2. WLC (7,1)
LCOPR00	1.0	1.1	1.6									1.6			1.0
LCOPR37	1.0	1.1		1.6									1.6		1.0
LCOPR90	1.0	1.1			1.6									1.6	1.0
LCSTM00	1.0	1.1				1.6			1.6						1.0
LCSTM37	1.0	1.1					1.6			1.6					1.0
LCSTM90	1.0	1.1						1.6			1.6				1.0

- Ensure that Analysis2 is active. The combinations must be put into this folder rather than the *Load Cases* folder. (The wave loads are found in Analysis2 and not *Load Cases*.)

- Use *Loads* | *Load Combination* to create the load combinations as shown below for LCOPR00. Double-click a factor to change it.



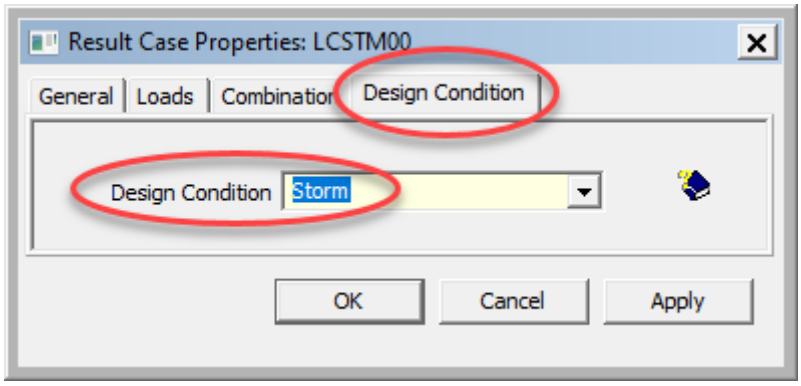
- Create the other five load combinations.

- Note that creating the first load combination is logged as:

```
LCOPR00 = LoadCombination(Analysis2);
LCOPR00.addCase(LCGrav, 1);
LCOPR00.addCase(LCEq, 1.1);
LCOPR00.addCase(Analysis2.WLC(4, 1), 1.6);
LCOPR00.addCase(Analysis2.WLC(7, 1), 1);
LCOPR00.addCase(WindOPR00, 1.6);
LCOPR00.convertLoadToMass = false;
LCOPR00.EquipmentRep = EquipmentAsLineLoads;
```

- Rather than creating all six combinations interactively, the above (except the command 'LCOPR00.convertLoadToMass = false;' which is superfluous) may be pasted into a file and edited to five sets of commands for creating the remaining five load combinations. These commands may then be pasted into the *Command Line* pane of GeniE.
  - This procedure is especially useful when there are tenths or hundreds of load combinations.

- The design condition must be set for the three storm load combinations. (This is relevant for code checking the analysis results.) This is done by right-clicking the load combination and going to the *Design Condition* tab as shown below.



- The above is logged as:

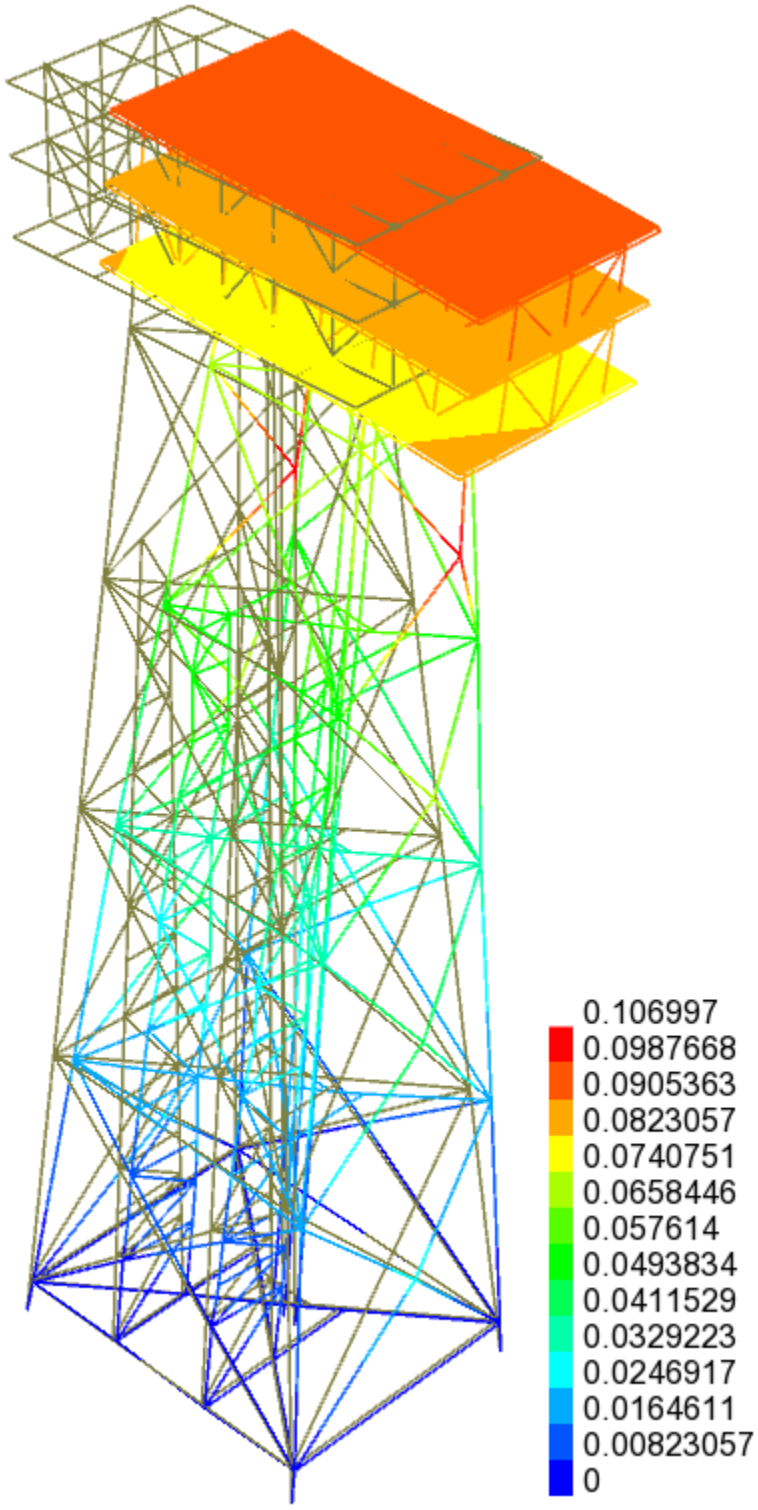
```
LCSTM00.designCondition = lcStorm;
```

- This command may be edited into the file with commands creating the load combinations.

7 RUN A STRUCTURAL ANALYSIS WITH WIND AND WAVE LOADS COMBINED

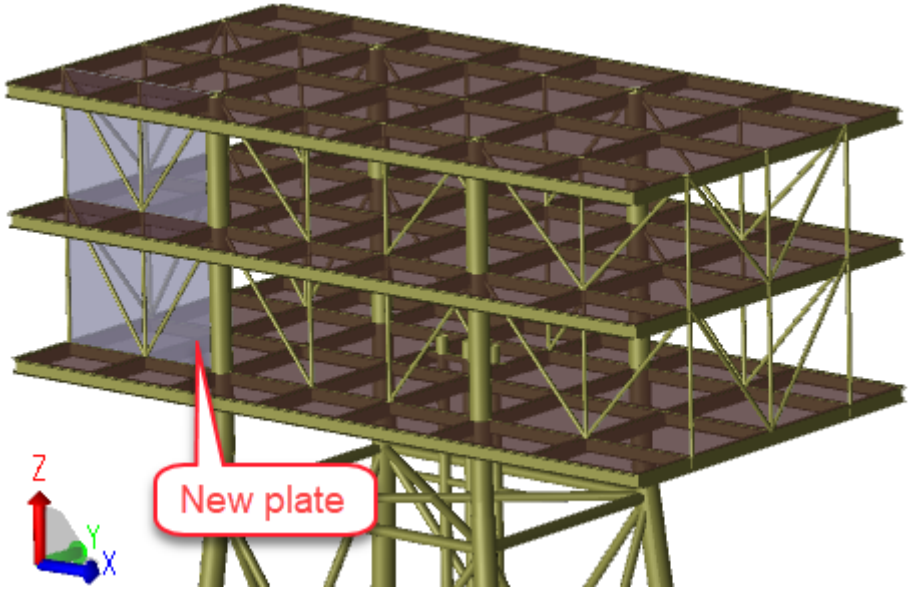
- With Analysis2 set as active use Alt+D to open the *Activity Monitor* and click *Start* to run the analysis.

- Switch to *Results - All* display configuration to view some results. Below displacements are colour coded on top of the deformed model together with the undeformed in wireframe mode.



8 WIND LOADS ON PLATES COMPUTED BY GENIE (NO ANALYSIS DONE)

- The imported model contains only deck plates but no walls that would be subjected to wind pressure, so to do an exercise with wind loads on plates create a **vertical plate** as follows:
  - First create the new plate shown.

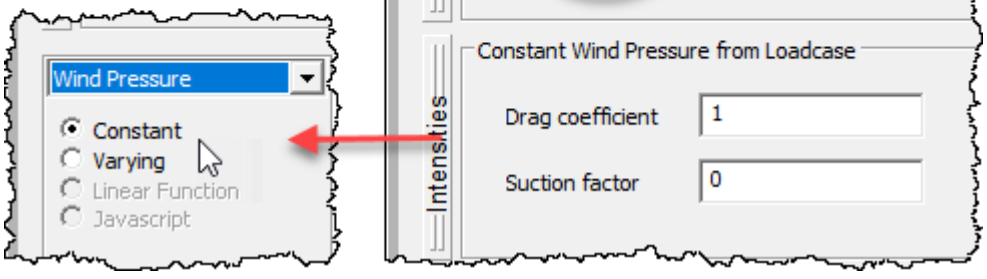


  - The plate shall have a small self weight and moderate stiffness so that it does not contribute to the stiffness of the topside.

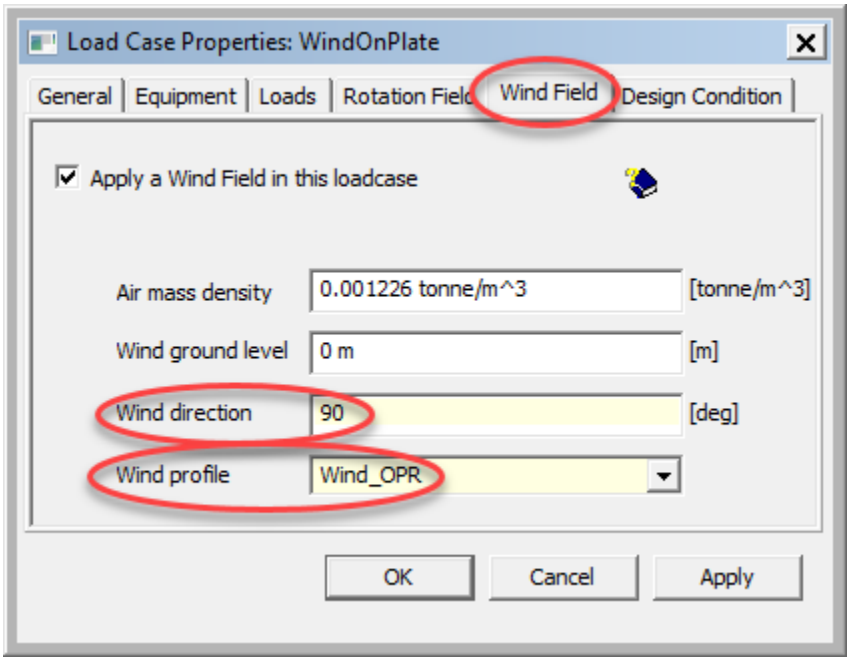
  - Note: The plate created shall NOT have the non-structural plate property assigned, as then no FEM mesh is created for the plate and no FEM loads will be generated.

- Create a new load case, named e.g. WindOnPlate, and make sure this is selected.

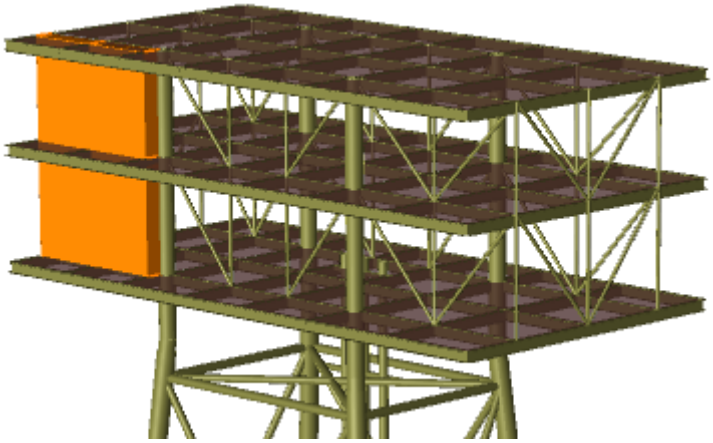
- Use *Loads* | *Explicit Load* | *Surface Load* to create a *Constant Wind Pressure* on the new plate.
  - Hover the mouse over the *Footprint* 'curtain' to select *Plate* Pl4.
  - *Front* is the bluish side of the plate.



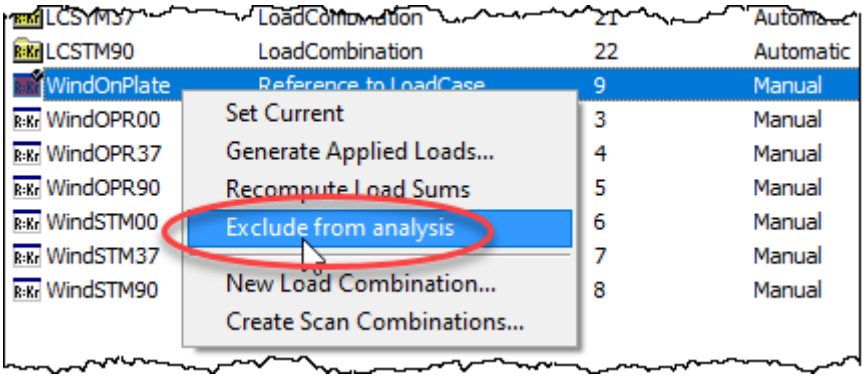
- Right-click the new load case and select *Properties*. In the *Wind Field* tab select wind direction 90 deg. and a wind profile.



- For this wind load case the conceptual load and applied load will appear equal in the display.



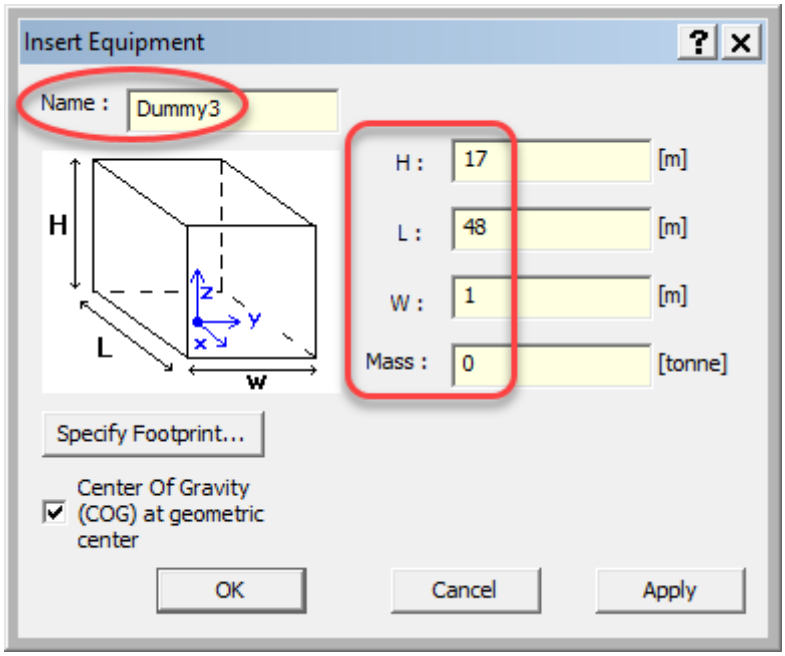
- As this wind load case is merely a small exercise on the side, we don't want to include it in an analysis.
  - Exclude it from the analysis Analysis2 by selecting it in the Analysis2 folder and excluding it as shown to the right. See that it disappears. (It is still in *Load Cases*.)



9 WIND LOADS AS POINT LOADS ON JOINTS (NO ANALYSIS DONE)

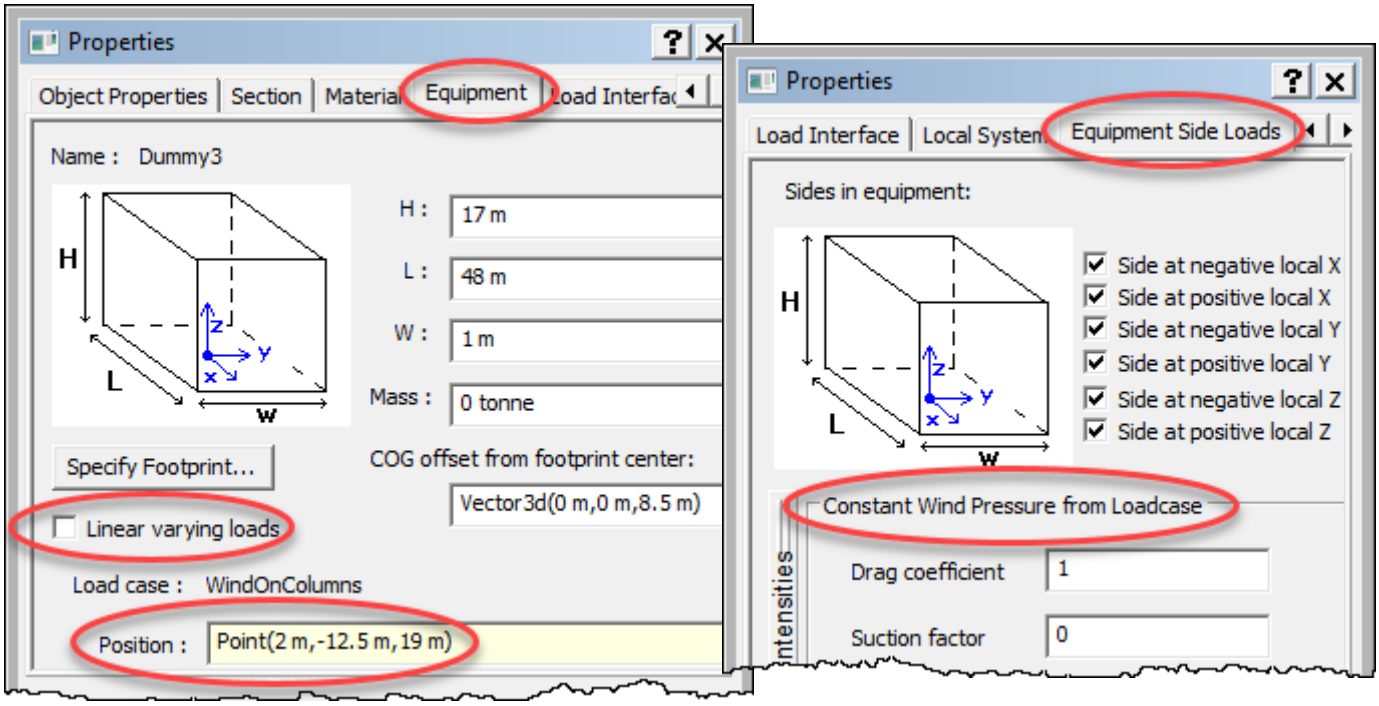
- We may want the wind loads on the whole topside structure to be applied directly to the main columns carrying the decks. To do so, follow the guideline below.

- Create the new dummy equipment representing the wind area of the whole topside structure for wind in direction 90 deg.



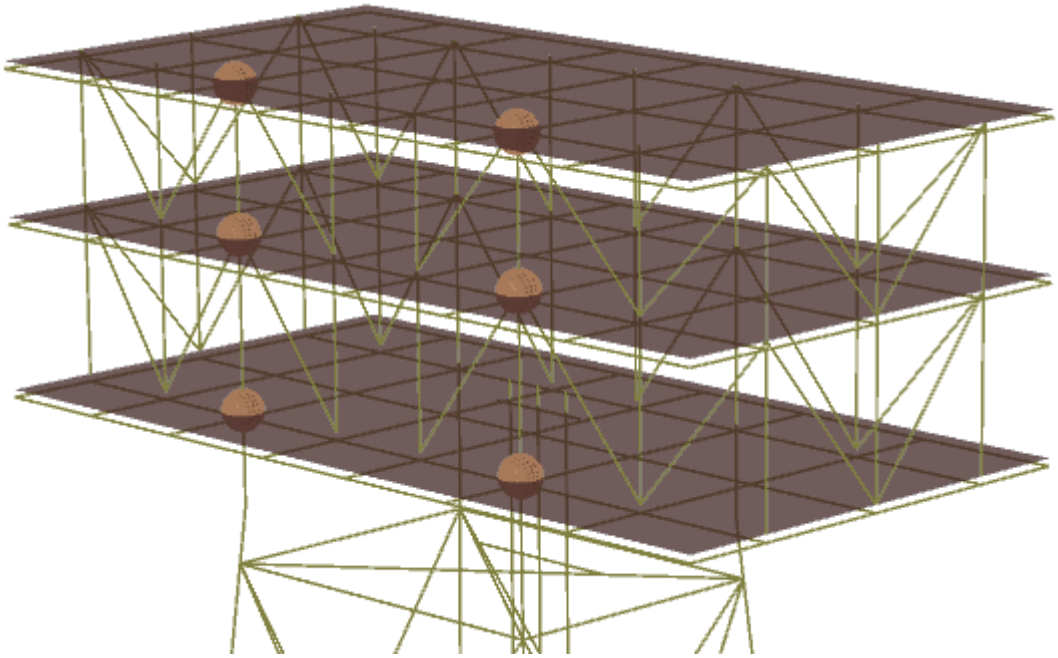
- Create a new load case, named e.g. WindOnColumns, and make sure this is selected.

- Place the equipment Dummy3 in this load case positioned as shown to the left below.
  - In the *Equipment Side Loads* tab select *Constant Wind Pressure*.

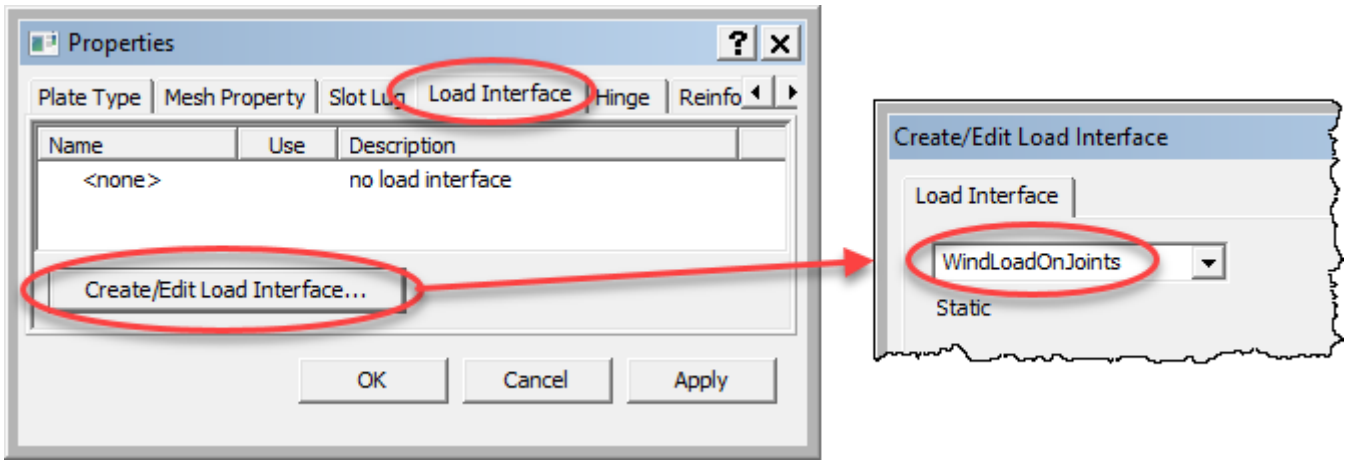


- To transfer the wind pressure on the equipment directly to the columns rather than to the girders supporting the equipment, two actions must be taken:
  - Create joints along the columns at positions where the wind load shall be applied.
  - Create a so-called load interface incorporating the equipment and the joints.

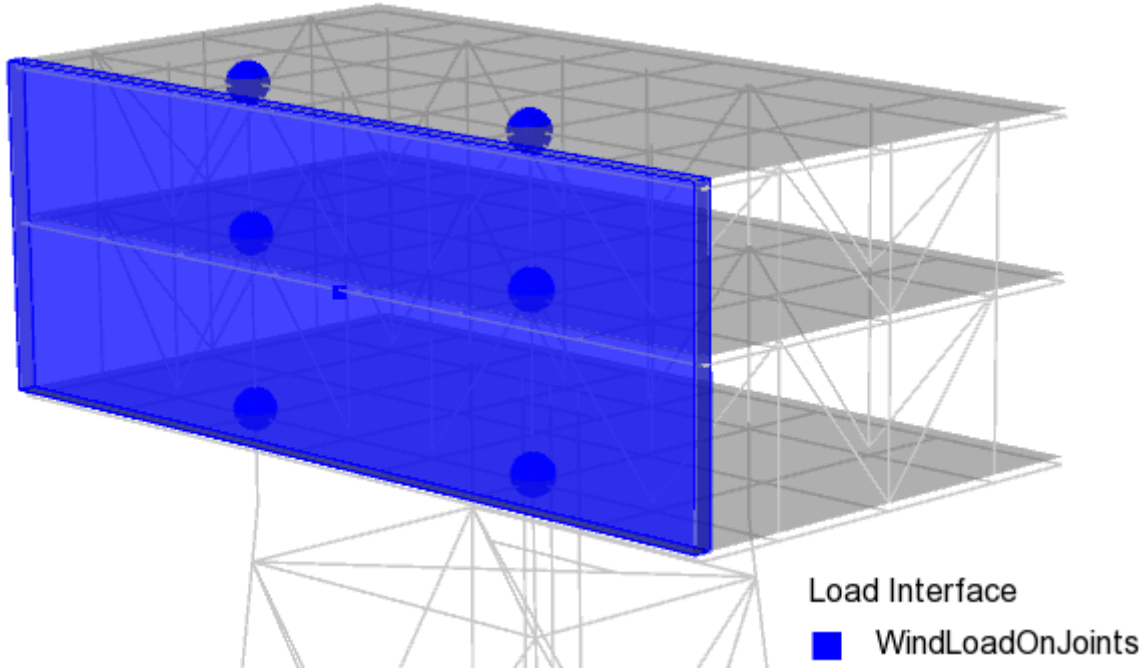
- Create the six joints shown below.



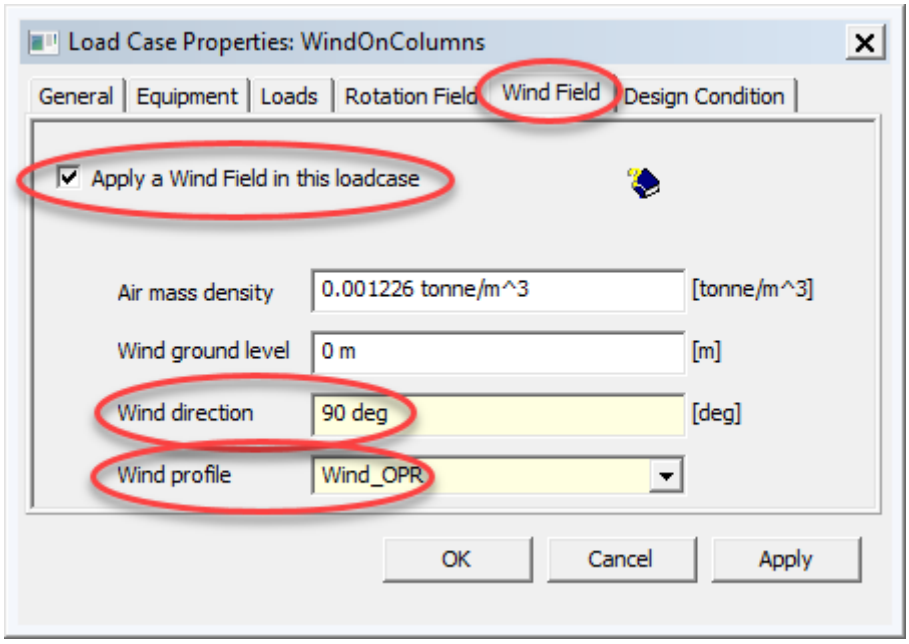
- Use *Edit* | *Properties* to open the *Properties* dialog. In the *Load Interface* tab click *Create/Edit Load Interface*. Give any name of the *Load Interface*, e.g. WindLoadOnJoints.



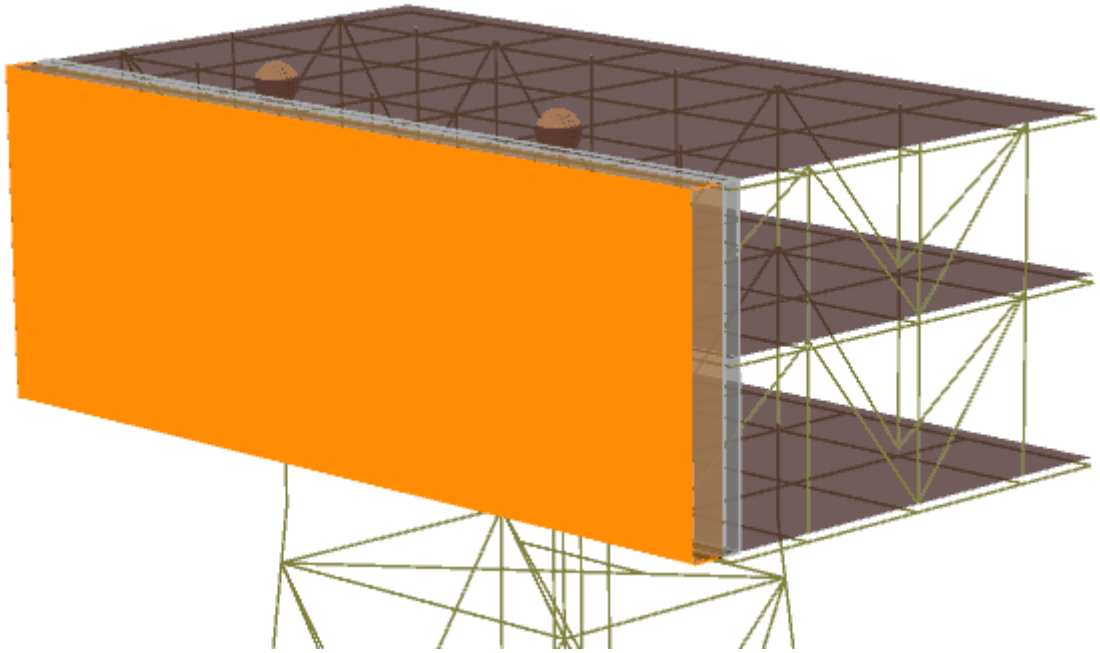
- Assign the load interface (found in the *Properties* | *Load Interfaces* folder in the browser) to both the equipment and joints. The load interface can be colour coded for verification.



- Select the load case in the *Load Cases* folder, right-click and select *Properties*. In the *Wind Field* tab specify wind direction 90 deg. (normal to the equipment) and select a *Wind profile*.

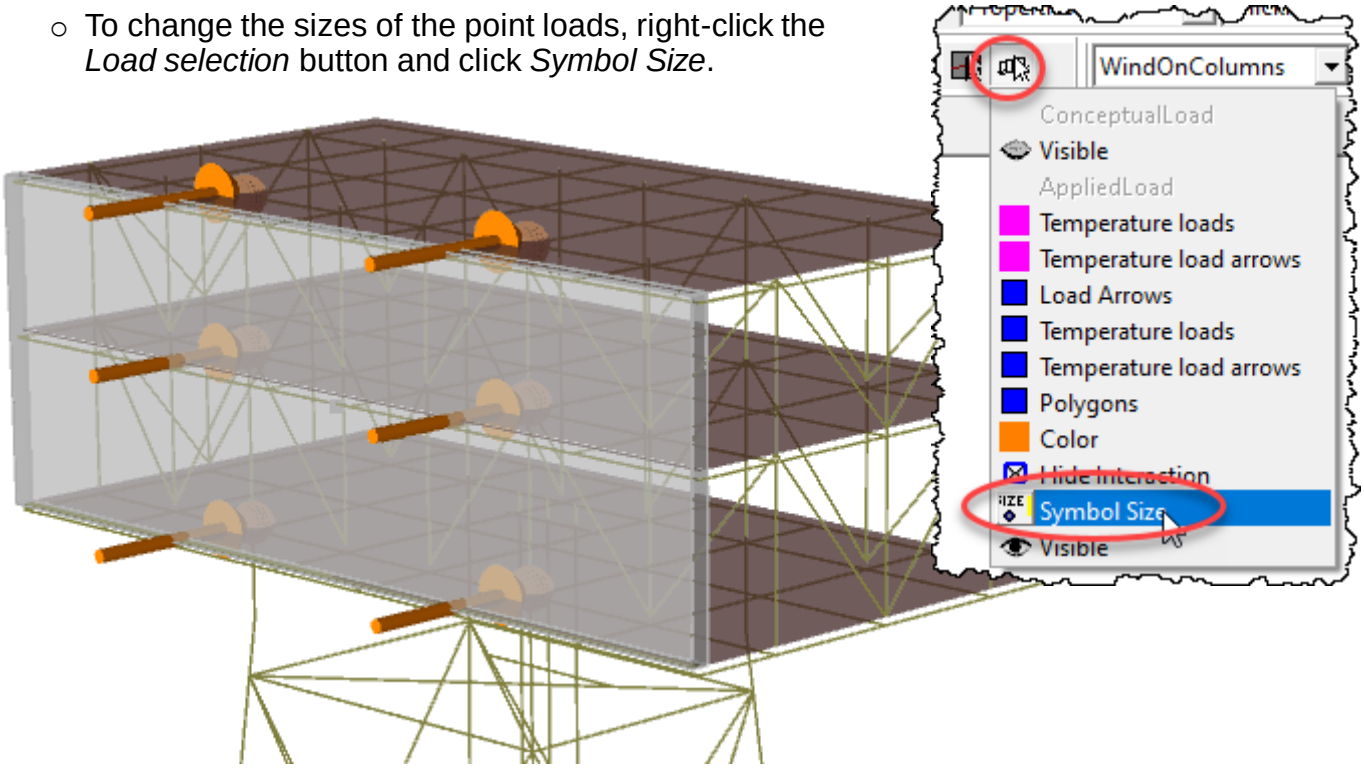


➢ The conceptual load is shown below.



➢ After generating applied loads (right-click the load case in the browser and select *Generate Applied Loads*) the applied loads (the loads used in the analysis) are displayed.

  - To change the sizes of the point loads, right-click the *Load selection* button and click *Symbol Size*.



➢ Exclude the load case WindOnColumns from Analysis2 as this wind load case is merely a small exercise on the side,

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