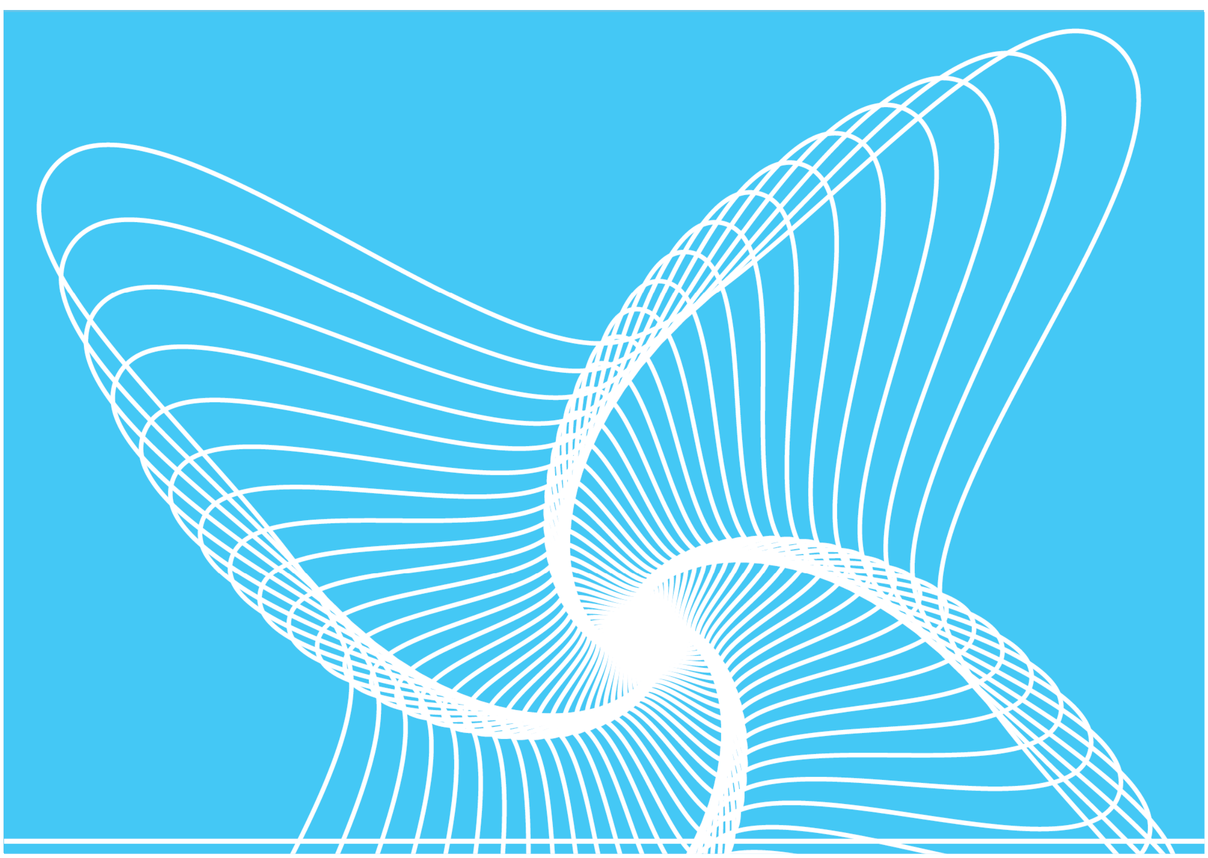
DNV·GL

SESAM TUTORIAL

HydroD

Stability and Hydrostatic Analysis of SemiSubmersible

Valid from program version 5.0-05, build 5196



SAFER, SMARTER, GREENER

Sesam Tutorial

HydroD – Stability and Hydrostatic Analysis of Semisubmersible

Date: 29 January 2016, Revision 03

Valid from HydroD version 5.0-05 build 5196

Prepared by DNV GL - Software

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

E-mail sales: software@dnvgl.com

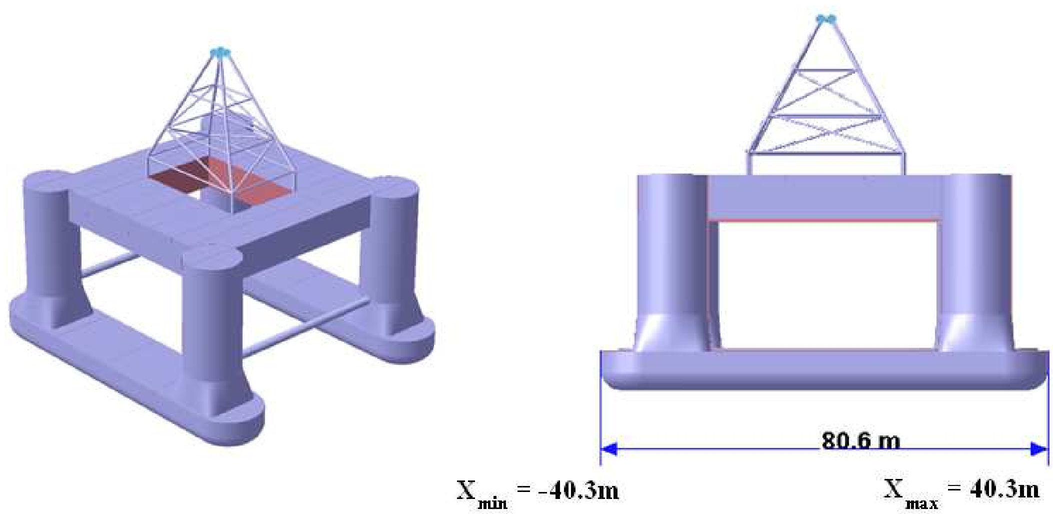
Table of contents

1 INTRODUCTION

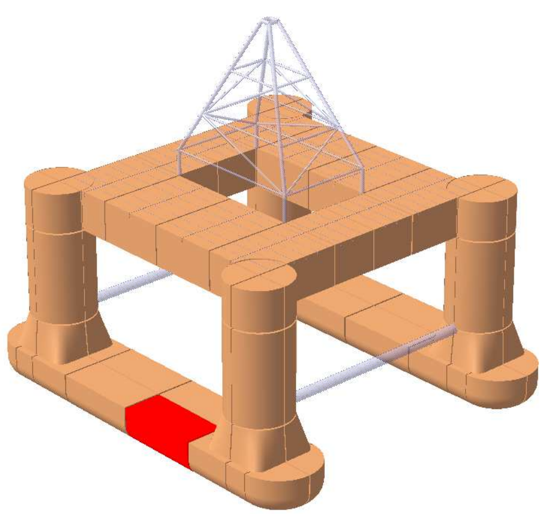
The text in this tutorial describes the necessary steps to do stability and hydrostatic analysis of a semi-submersible as well as checking the compliance to NMA stability rule.

The semi-submersible has 22 compartments which can be filled independently – in this tutorial you are asked to do various tank fillings to see the effect on stability parameters like e.g. equilibrium position and the computed GZ-curves.

The panel model, compartment model (used to describe the tanks) and the mass model have been created in GeniE. The total mass of the structure (mass model) is 21258989.9 kg. The file describing the panel model is called Panel_T543.FEM (notice that this file uses a Panel_T1.FEM file), the compartment model and the mass model is named Structure_T4.FEM.



The model created in GeniE



The compartment model and mass model defined in GeniE

In addition to the text input, there is also a journal file Semi_submersible_Stability.js that you can read into HydroD to rapidly reproduce the workshop. Please make sure that the journal file and the FEM files are in the same folder.

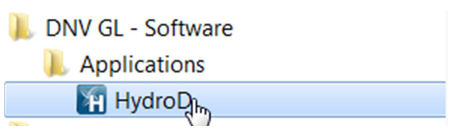
This tutorial should be viewed on-line or on colour print out to best see the property colour coding.

2 START HYDROD AND MAKE A NEW WORKSPACE

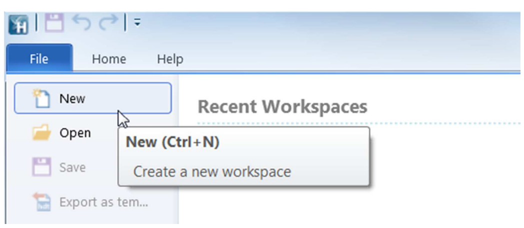
You start HydroD from

- Desktop

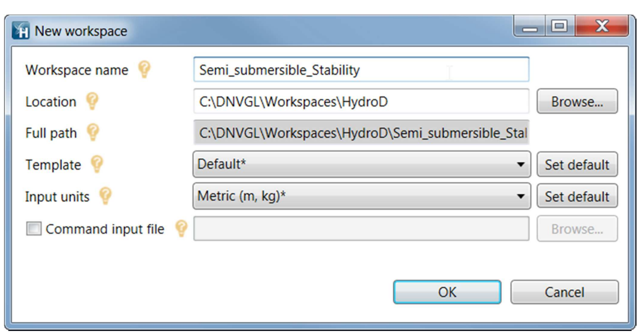
- Start All Programs |DNV GL - Software\Applications |HydroD



You make a new workspace from Ribbon button bar

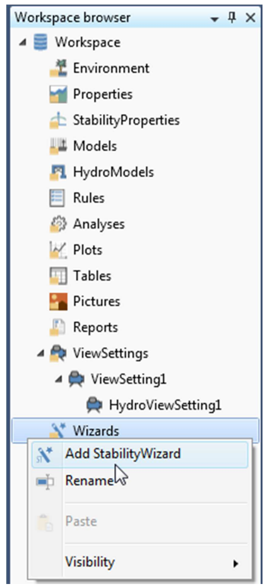


- For this workspace, we use input units – Metric (m, kg)
- In this tutorial we have used the default Workspace directory which is C:\DNVGL\Workspaces\HydroD. You may of course use other directories.

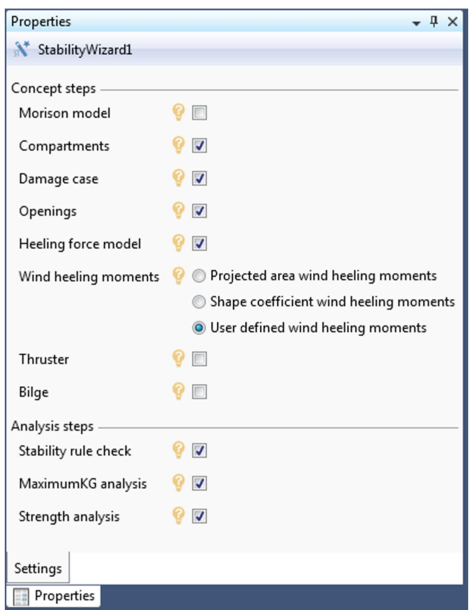


3 STABILITY WIZARD

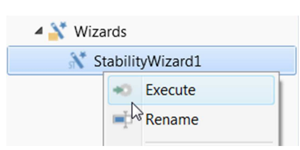
Add a *StabilityWizard* to the *Wizards* folder in the *Workspace browser*.



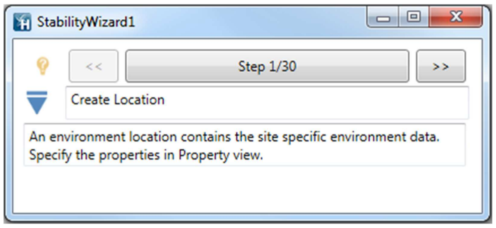
Go to the *Properties* window of *StabilityWizard1* and set the properties as below



Then execute *StabilityWizard1*. A wizard dialog will pop up to guild you to set up the stability analyses step by step.



Step 1 – Create Location

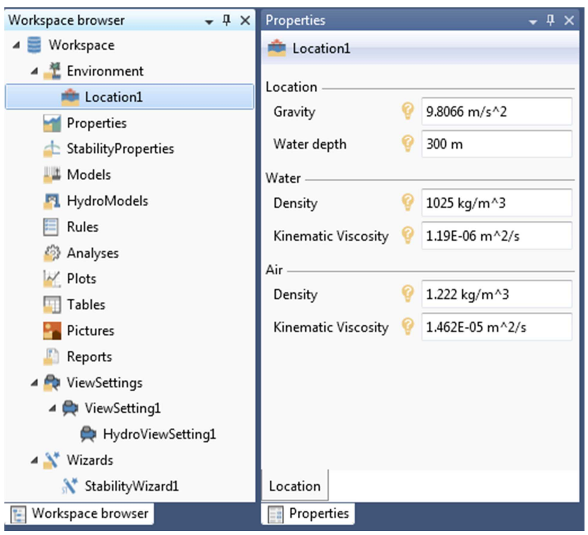


In the wizard dialog, there are three buttons on the top:

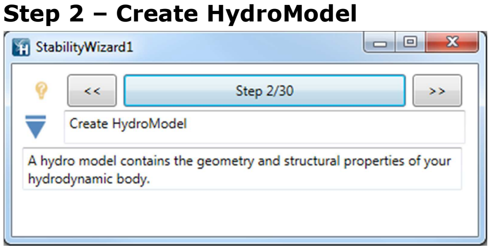
- *Step backward*
- *Step*
- *Step forward*

You could use the *Step backward* and *Step forward* to navigate to a backward or forward step for changes. When you click the *Step* button in the middle, corresponding concept will be created (the first time you click it) and selected in the *Workspace browser* and its properties will show in the *Properties* window.

In this step, a *Location* is created and you could change its properties of the *Loaction* in the *Properties* window. In this tutorial, we keep the default values.

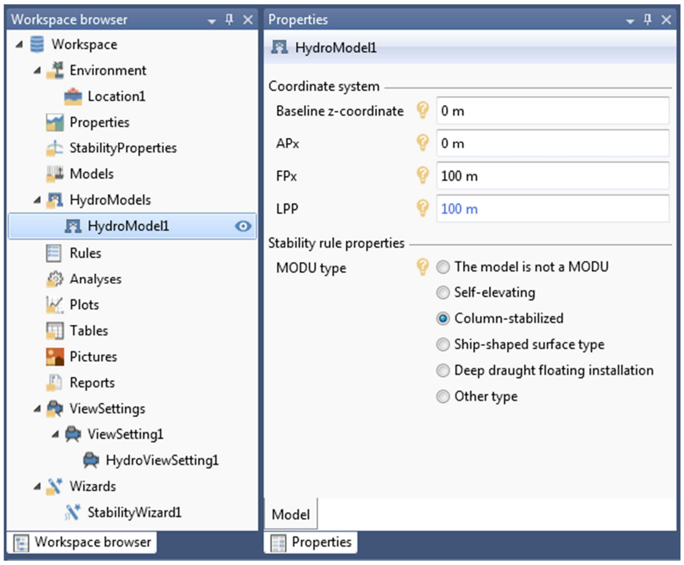


Step 2 – Create HydroModel

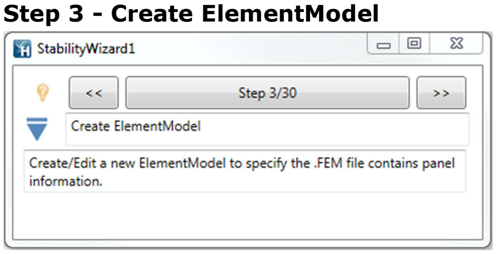


In this step, a *HydroModel* will be created, which contains the geometry and structural properties of the semi-submersible.

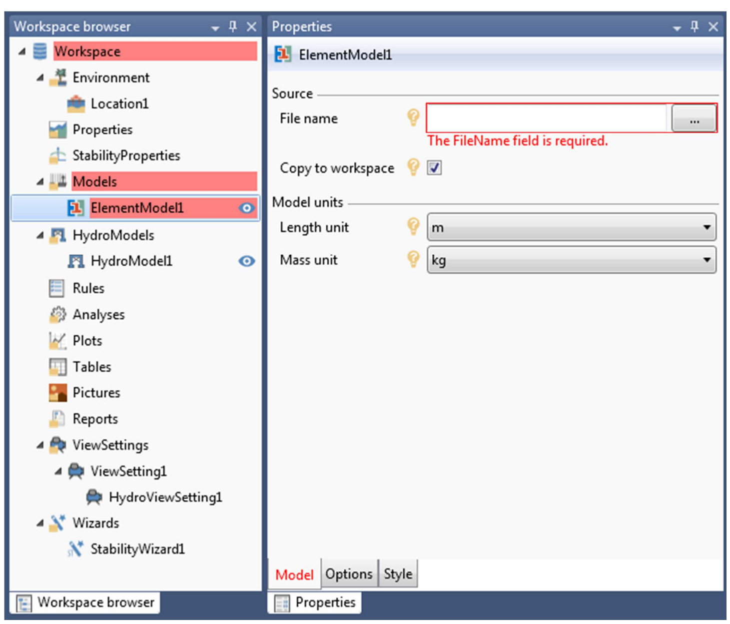
In the tutorial, we specify the *MODU type* as *Column-stabilized* in the *Properties* window.



Step 3 - Create ElementModel

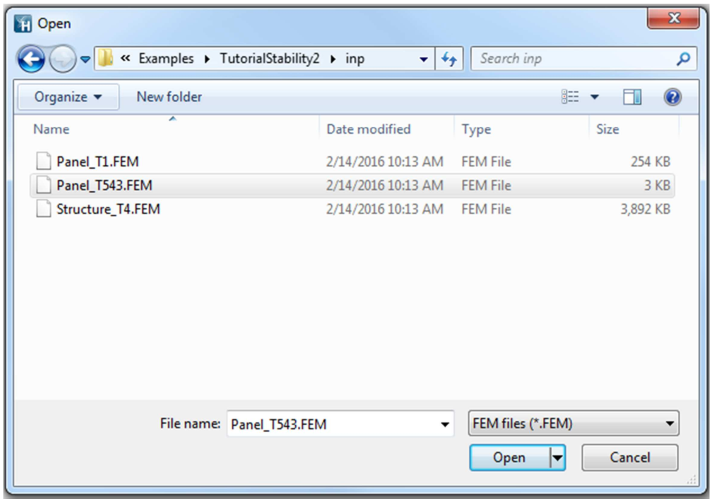


In this step, an *ElementModel* will be created, which will be referred by the PanelModel in next step.

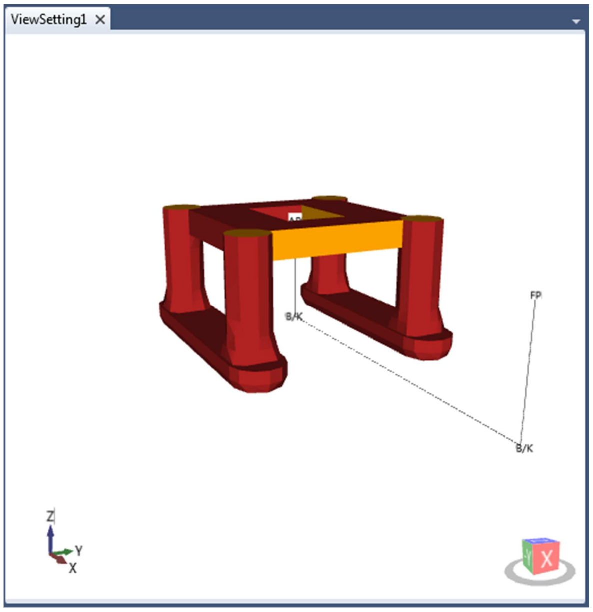


The created *ElementModel* is marked as red in the *Workspace browser*. This is because the *file name* is not specified yet. Such error information could also be found in the *Status* window.

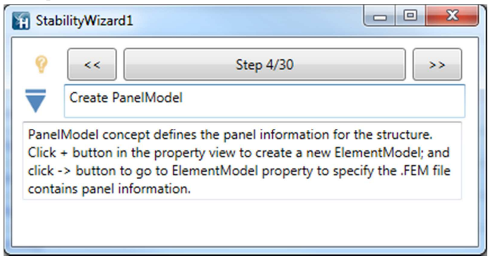
Locate the file Panel_T543.FEM stored under C:\Program Files\DNVGL\HydroD V5.0-05\Doc\Examples\TutorialStability2\inp. The path name assumes you have installed the program HydroD using default values when installing.
Remember that this file refers to Panel_T1.FEM.



After this step, the *ElementModel* is shown in the 3D window.



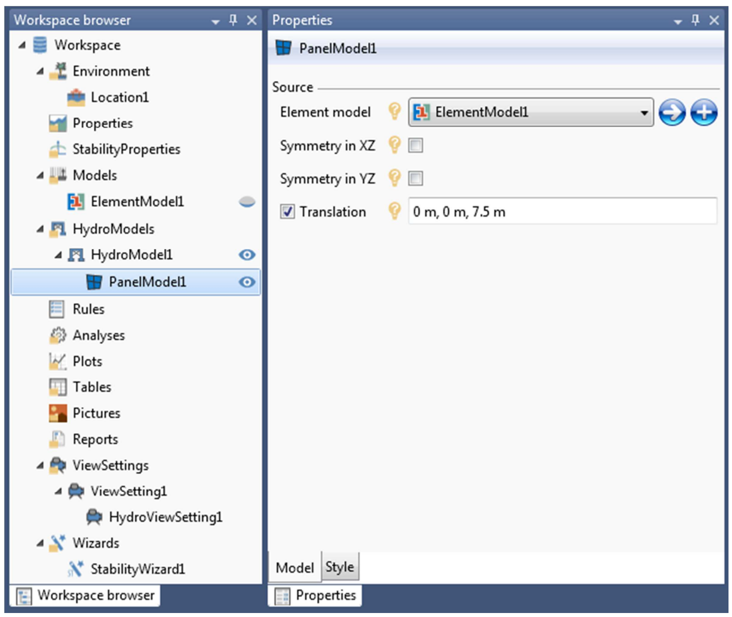
Step 4 – Create PanelModel



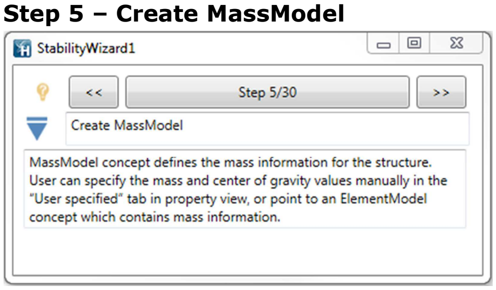
In this step, a *PanelModel* will be created, which specifies the outer wetted surface to calculate the floater buoyancy.

The created *PanelModel* is referred to *ElementModel1* created in previous step.

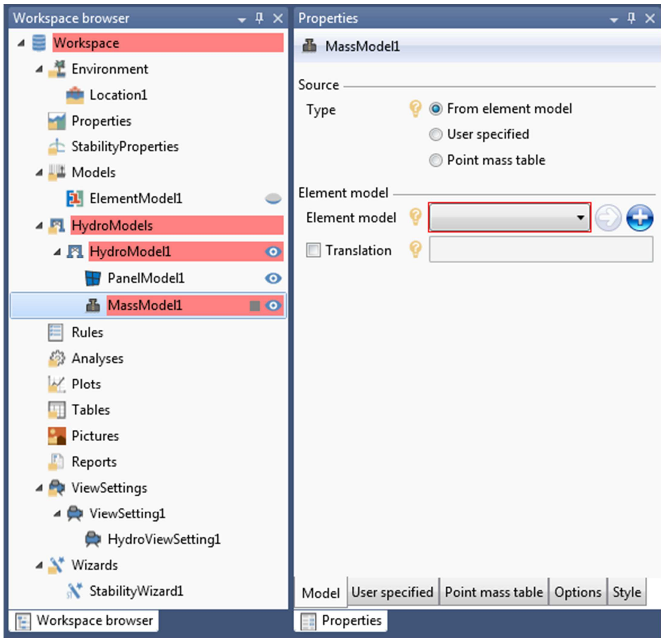
There are no symmetry planes in this model and a translation of 7.5 m in z direction is applied.



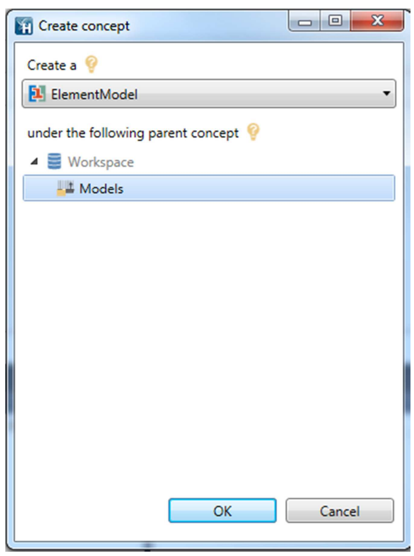
Step 5 – Create MassModel



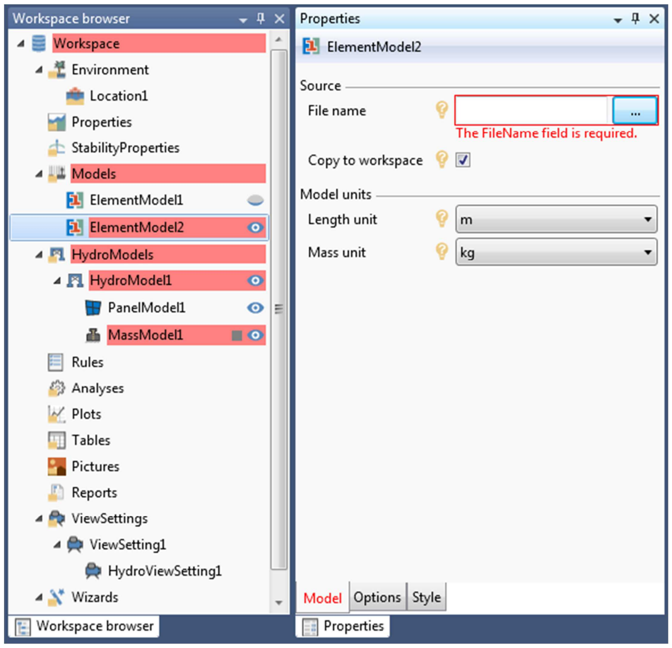
MassModel may be from an *element model*, specified by user or defined by point mass table. In this tutorial, the mass model is from an element model.



Click the "+" button to add an *ElementModel* to the *Models* folder in the *Workspace browser*.

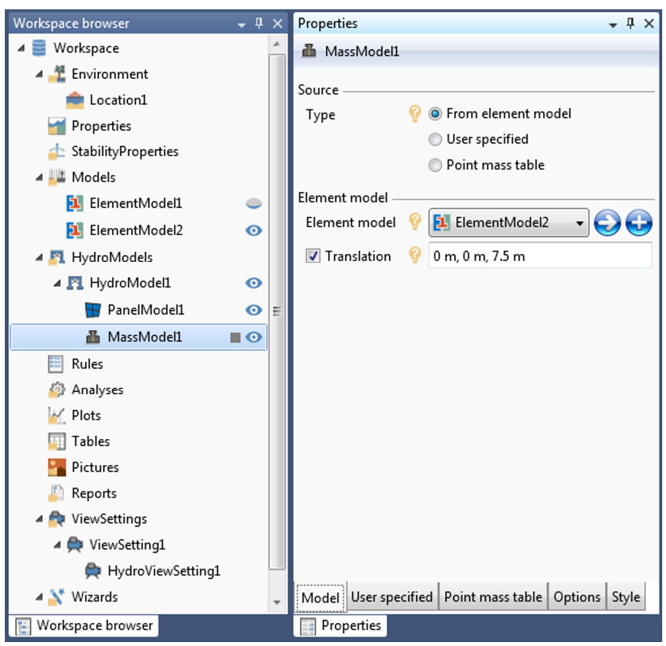


Then click the "->" button to navigate to the *Properties* window of *ElementModel*.

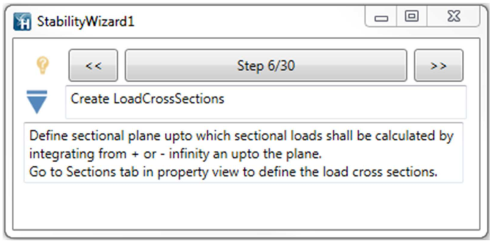


Locate the file Structure_T4.FEM stored under C:\Program Files\DNVGL\HydroD V5.0-05\Doc\Examples\TutorialStability2\inp. The path name assumes you have installed the program HydroD using default values when installing.

Set a translation of 7.5 m in z direction in the properties view of the *MassModel*.



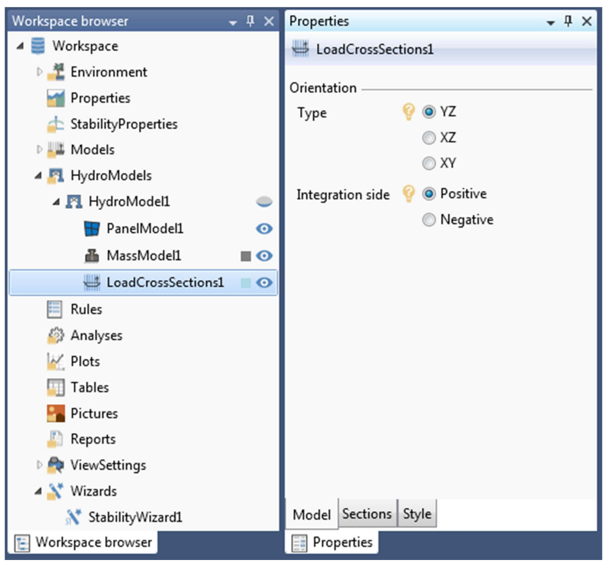
Step 6 – Create LoadCrossSections

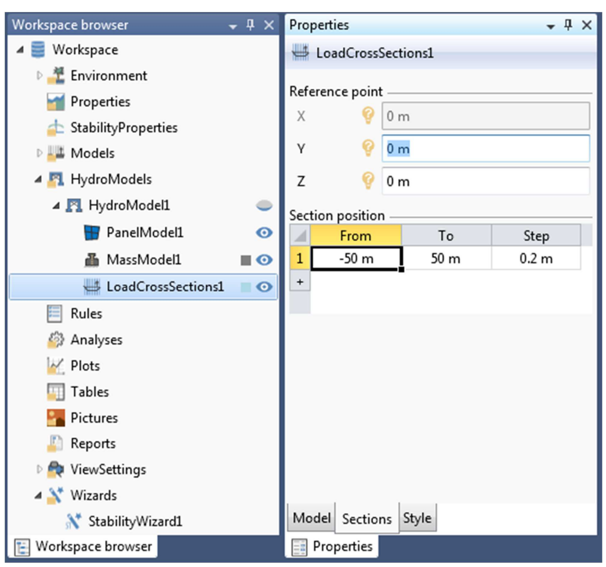


To compute still water forces and moments, you need to specify at which cross sections the program shall compute these.

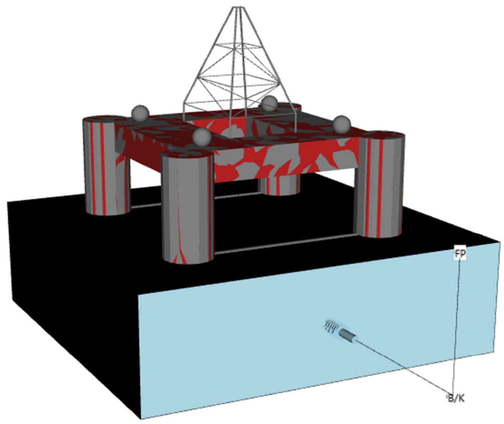
In the *Model* tab of *Properties* window, specify *Type* as *YZ* and *Integration side* as *Positive*.

In the *Sections* tab of *Properties* window, set the section positions as below.

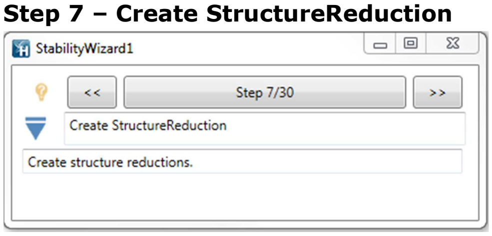




The cross sections are shown in the 3D window.



Step 7 – Create StructureReduction

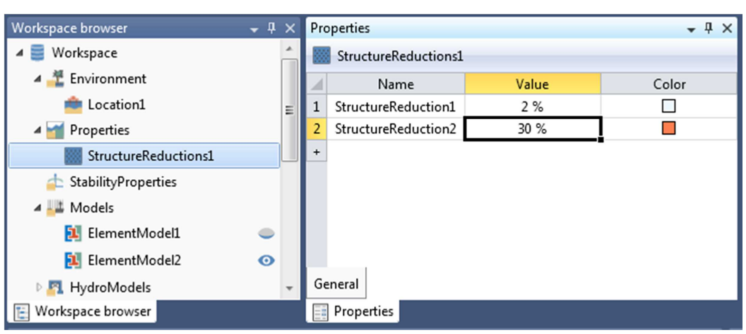


StructureReduction describes the fraction of the compartment volume that is occupied by structure or devices. The fraction of the compartment volume that can be filled with fluid in intact condition equals to 100% minus value of *StructureReduction*.

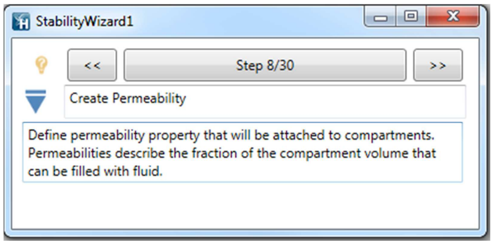
In this tutorial, define the *StructureReductions* as below.

Name	Value
StructureReduction1	2%
StructureReduction2	30%

By clicking the *Step* button, *StructureReductions1* and *StructureReduction1* will be created automatically. And *StructureReduction2* can be created by clicking the "+" button in the *properties* window of *StructureReductions1.*



Step 8 – Create Permeability

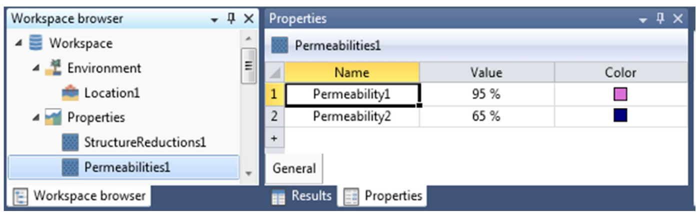


Permeability describes the fraction of the compartment volume that can be filled with sea water in damaged condition.

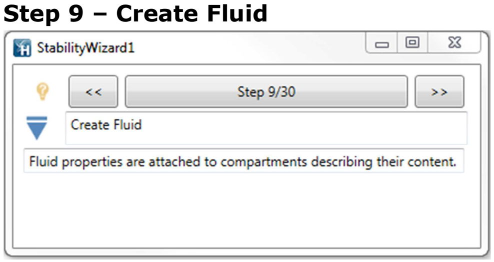
In this tutorial, define the *Permeabilities* as below.

Name	Value
Permeability1	95%
Permeability2	65%

By clicking the *Step* button, *Permeabilities1* and *Permeability1* will be created automatically. And *Permeability2* can be created by clicking the "+" button in the *properties* window of *Permeabilities1.*

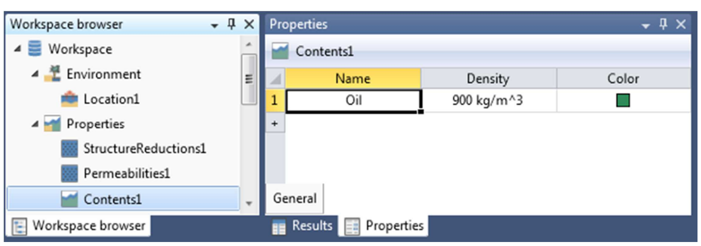


Step 9 – Create Fluid

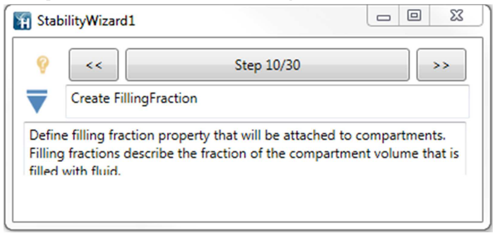


Fluid describes the intact fluid density in compartments.

By clicking the *Step* button, *Contents1* and *Fluid1* will be created automatically. Change the density to 900 kg/m^3 and rename it to *Oil*.



Step 10 – Create FillingFraction

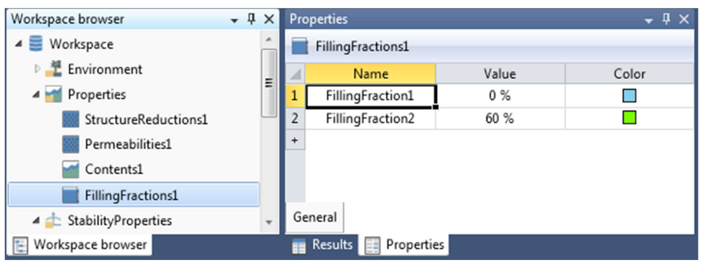


FillingFraction is used to specify how much a compartment is filled in intact condition, which may vary from 0% – 100%.

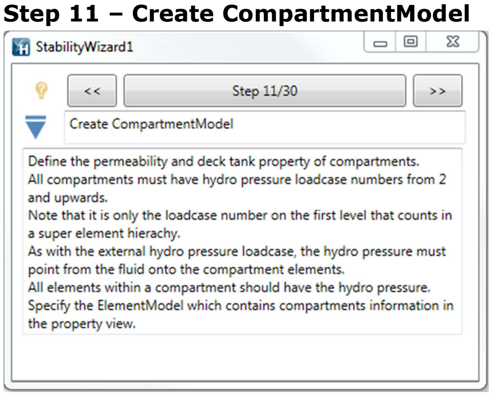
In this tutorial, define the *FillingFraction*s as below.

Name	Filling Fraction
FillingFraction1	0%
FillingFraction2	60%

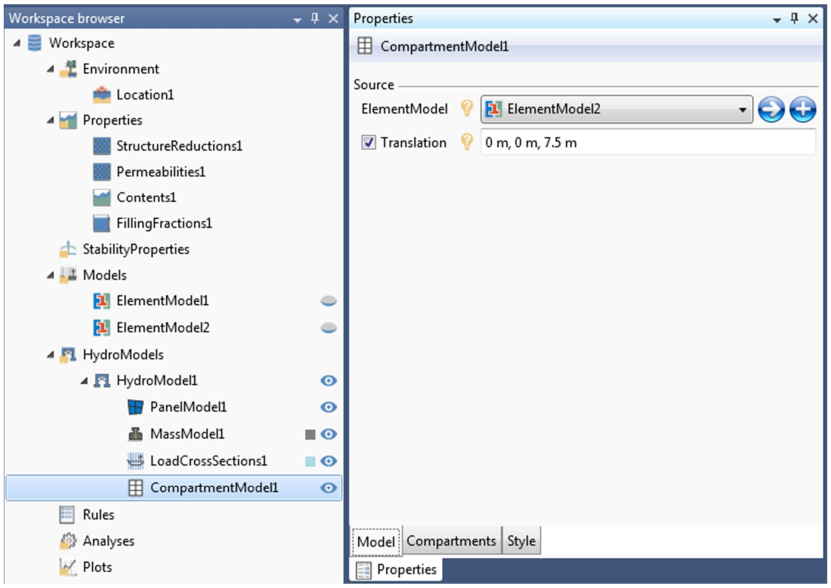
By clicking the *Step* button, *FillingFractions1* and *FillingFraction*1 will be created automatically. And *FillingFraction*2 can be created by clicking the "+" button in the *properties* window of *FillingFractions*1*.*



Step 11 – Create CompartmentModel

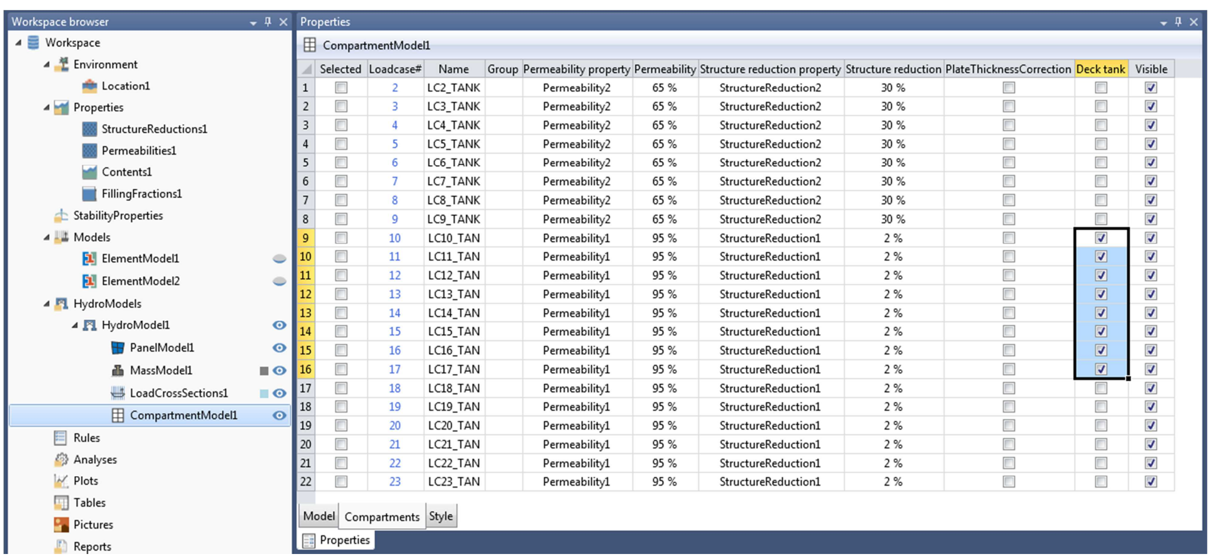


Similar to *PanelModel*, the model of compartments has been created in GeniE. In this tutorial, *EelementModel2* is used for the *CompartmentModel*. A translation of 7.5 m in z direction is applied.

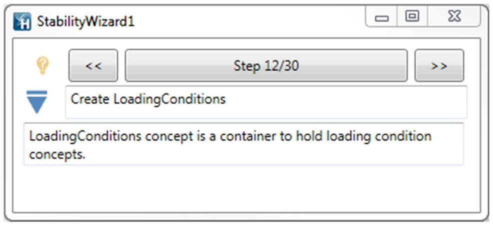


All the compartments defined in Structure_T4.FEM will be listed in the *Compartments* tab of *Properties* window.

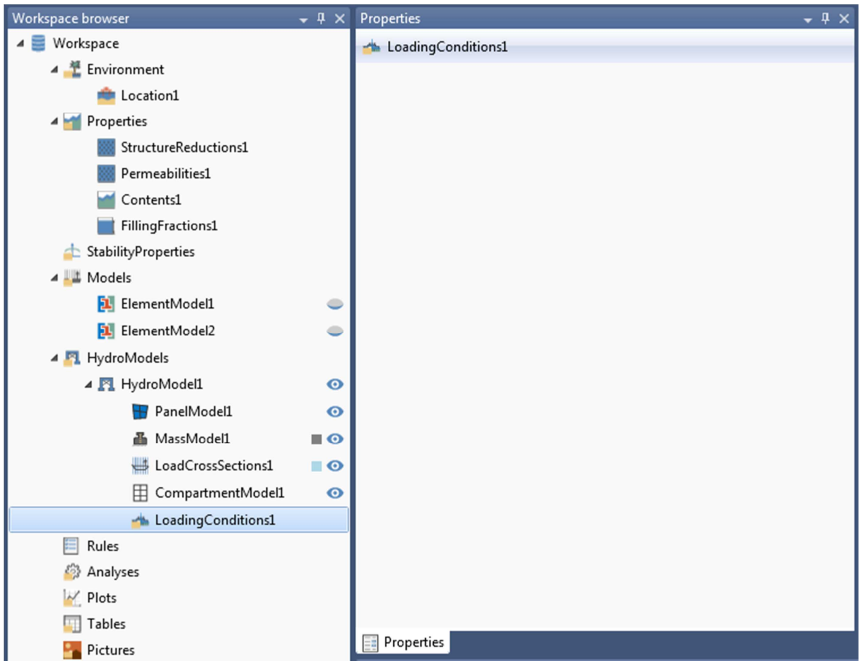
Define permeability and structure reduction of compartments as below and set compartments 10-17 as deck tanks.



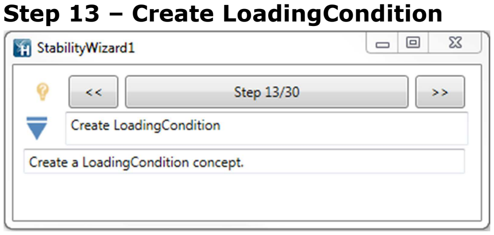
Step 12 – Create LoadingConditions



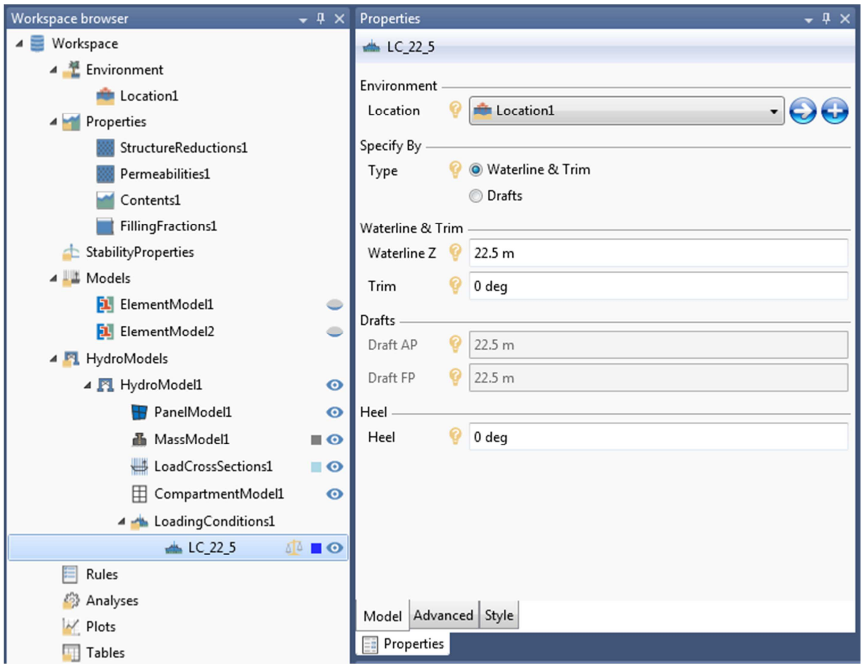
LoadingConditions is a container to hold a group of *LoadingCondition*. After it is created, we move to the next step directly.



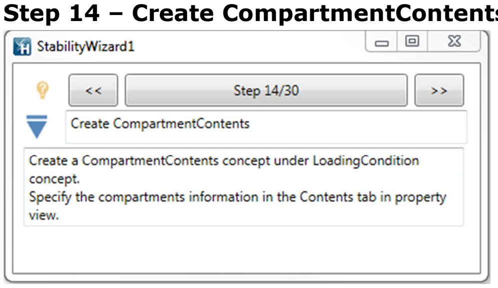
Step 13 – Create LoadingCondition



Add a *LoadingCondition* to the *LoadingConditions* which is created in the previous step, set *Waterline Z* to 22.5 m and rename the *LoadingCondition* to *LC_22_5*.

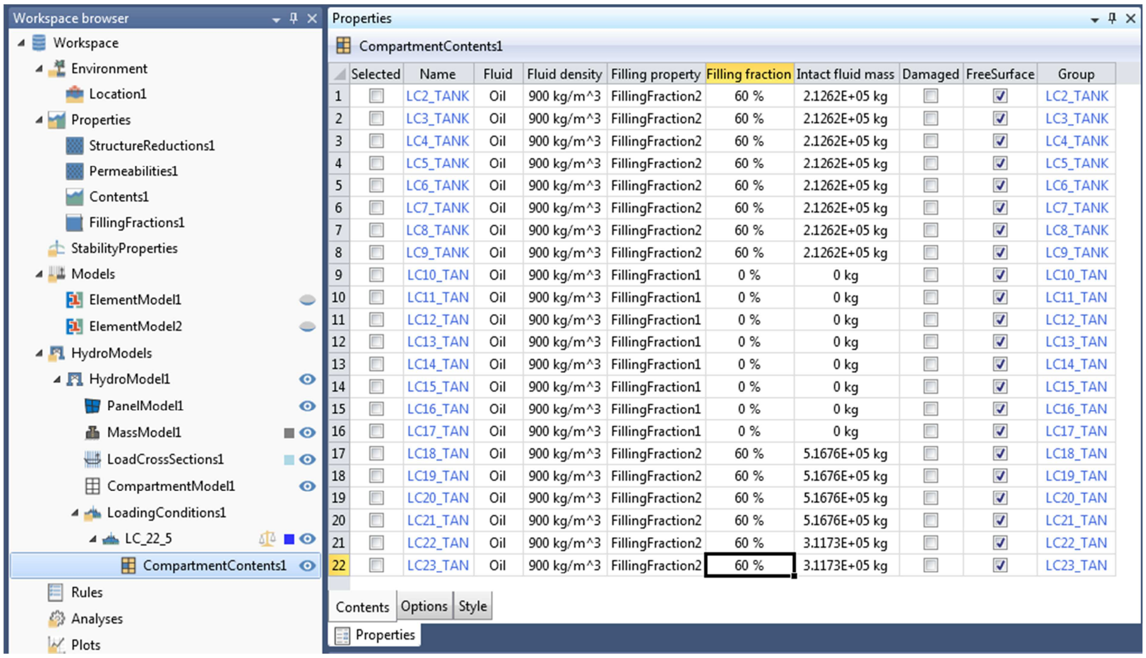


Step 14 – Create CompartmentContents



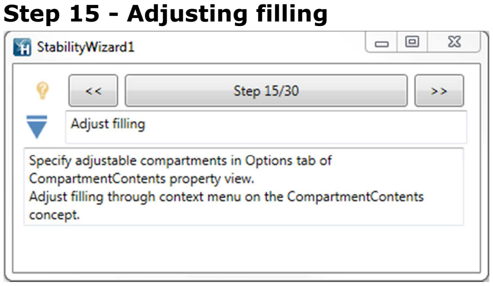
CompartmentContents defines the fluid content of each compartment for a certain *LoadingCondition*.

Define *Fluid* and *Filling fraction* for each compartment as below.



A symbol of "unbalanced" shows on the right of *LC_22_5* in the *Workspace browser*, indicating the loading condition is not in equilibrium.

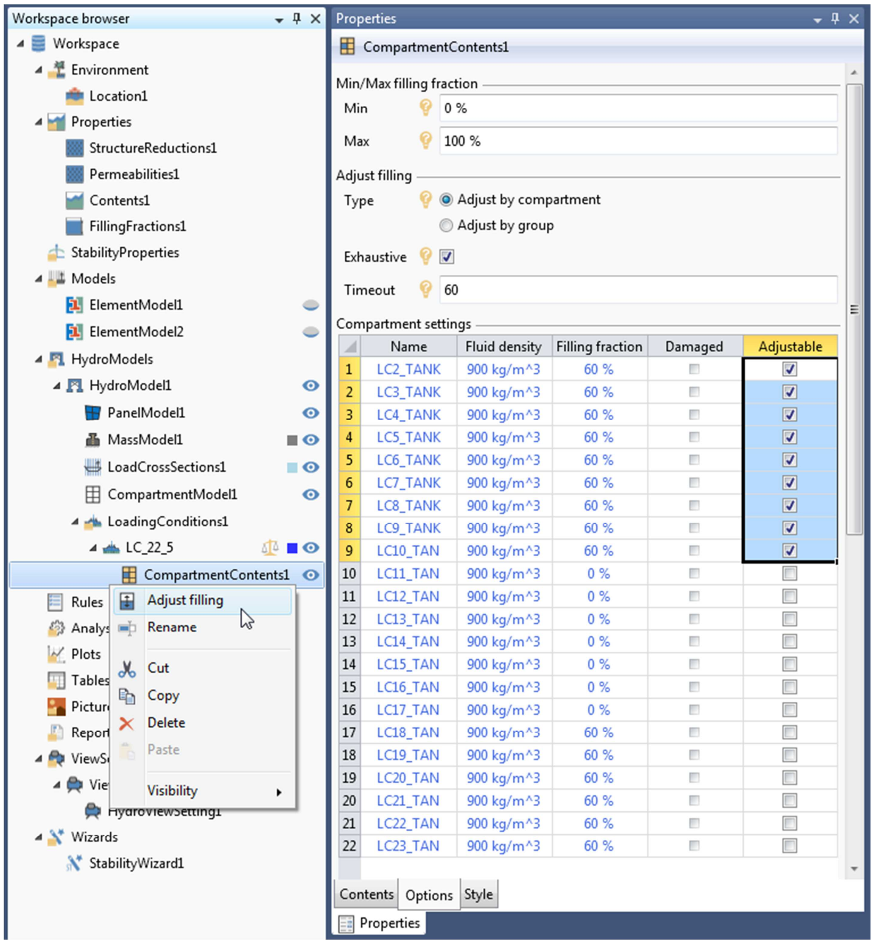
Step 15 - Adjusting filling



As *LC_22_5* is not in equilibrium, this step is to adjust the filling fraction of compartments.

By clicking the *Step* button, the *Properties* window - *Options* tab of *CompartmentContents* is activated. Select "Adjust b*y compartment"* and set *Adjustable* for compartments 2~10.

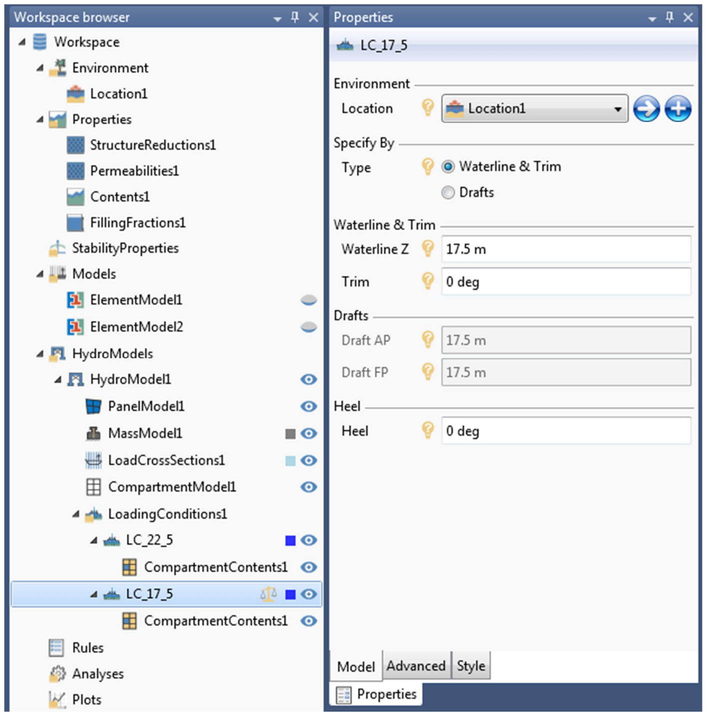
In the context menu of *CompartmentContents*, run the command *Adjusting filling*. The filling fraction of adjustable compartments will be adjusted automatically, so that *LC_22_5* is in equilibrium.



Before we go to next step, we add one more *LoadingCondition*.

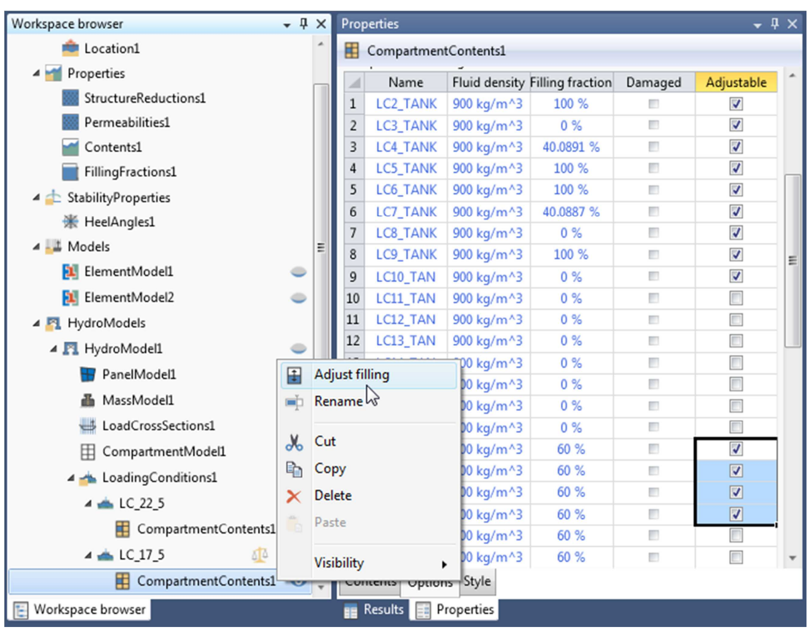
Copy *LC_22_5* and paste it to the parent folder *LoadingConditions1*

Change *Waterline Z* of the pasted *LoadingCondition* to 17.5 m and rename it to *LC_17_5*.

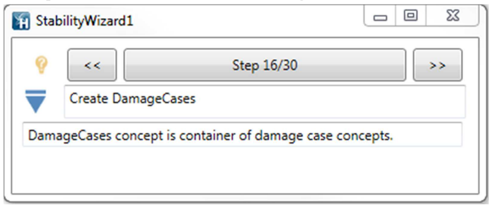


For *LC_17_5*, set *Adjustable* for compartments 2~10 and 18~21.

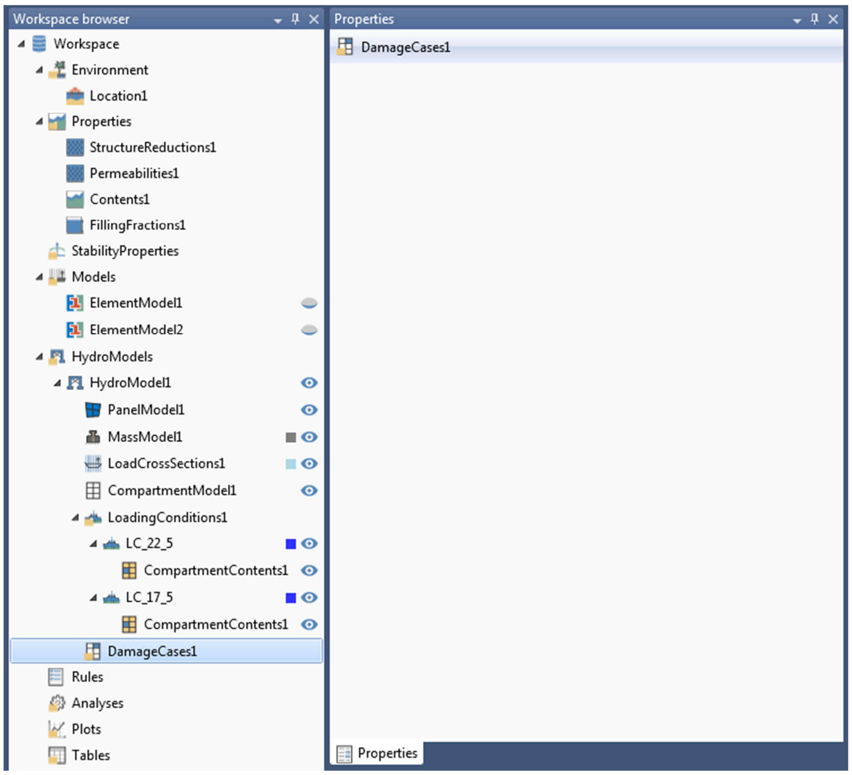
In the context menu of *CompartmentContents*, run the command *Adjusting filling*. The filling fraction of adjustable compartments will be adjusted automatically, so that *LC_17_5* is in equilibrium.



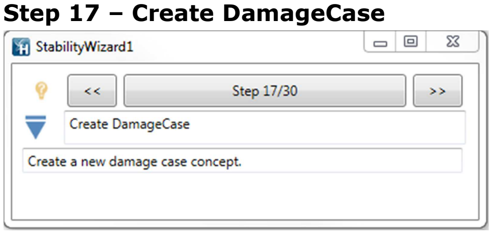
Step 16 – Create DamageCases



DamageCases is a container to hold a group of *DamageCase*. After it is created, we move to the next step directly.

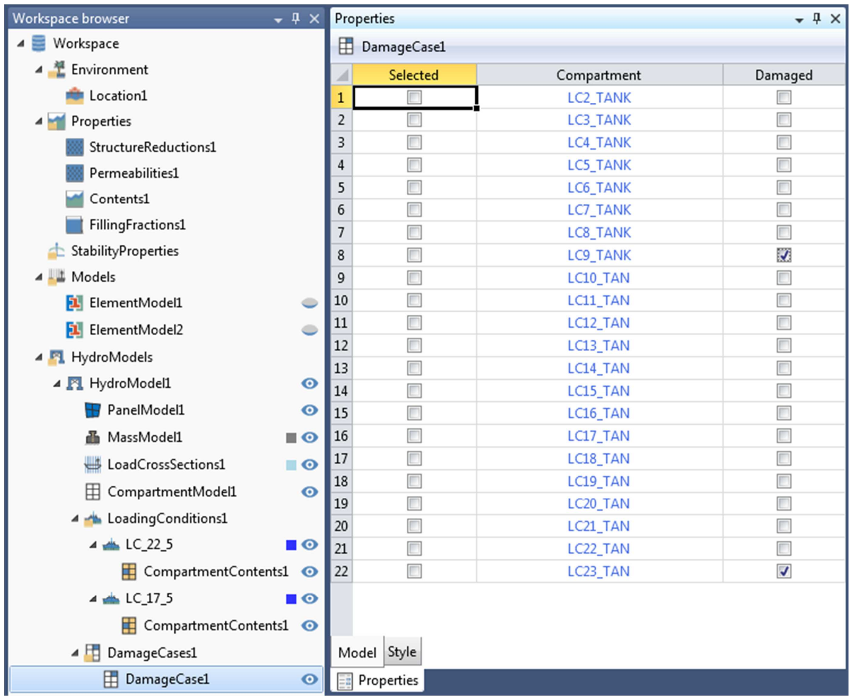


Step 17 – Create DamageCase

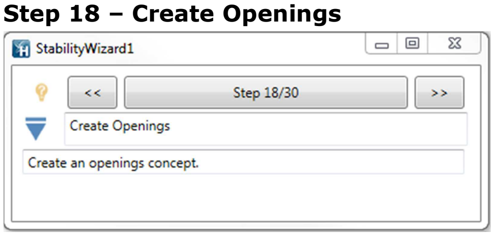


DamageCase defines a damage case for the compartments.

In the tutorial, we set *LC9_TANK* and *LC23_TANK* as damaged.



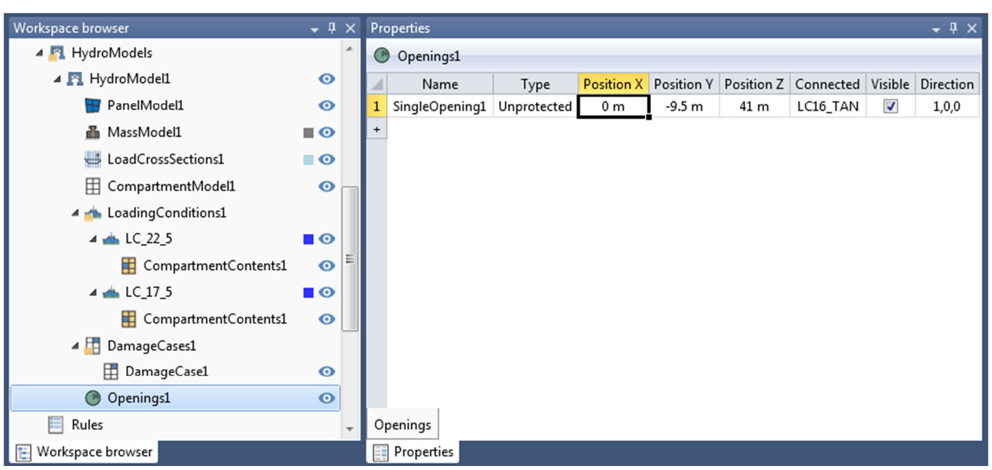
Step 18 – Create Openings



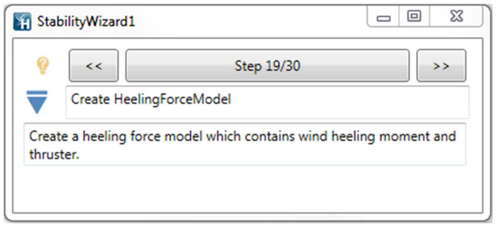
Flooding openings are connected to compartment. If an opening is submerged, the connected compartment will be flooded if the opening is not watertight.

In this tutorial, an *Opening* is defined at point (0 m, -9.5 m, 41 m) and connected to LC16_TANK.

By clicking the *Step* button, *Openings1* and *SingleOpening1* will be created automatically.

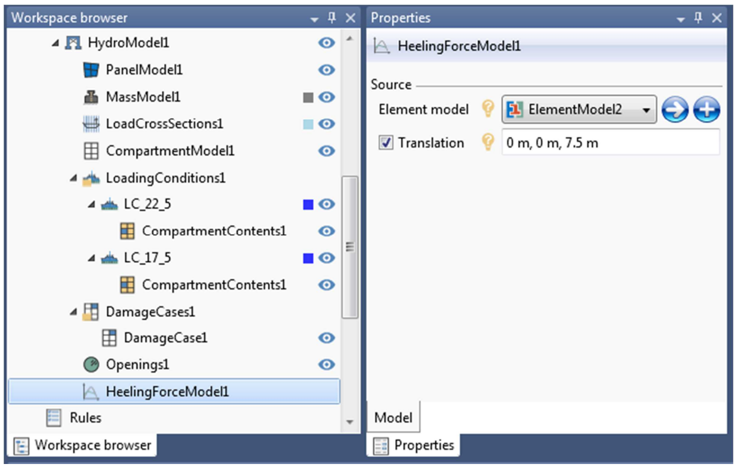


Step 19 – Create HeelingForceModel

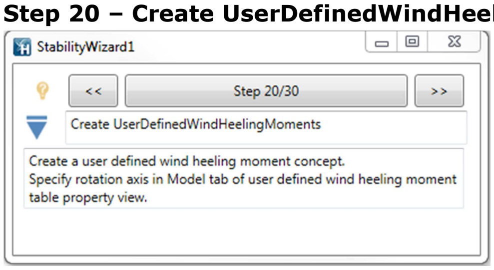


HeelingForceModel defines the heeling forces for stability analysis.

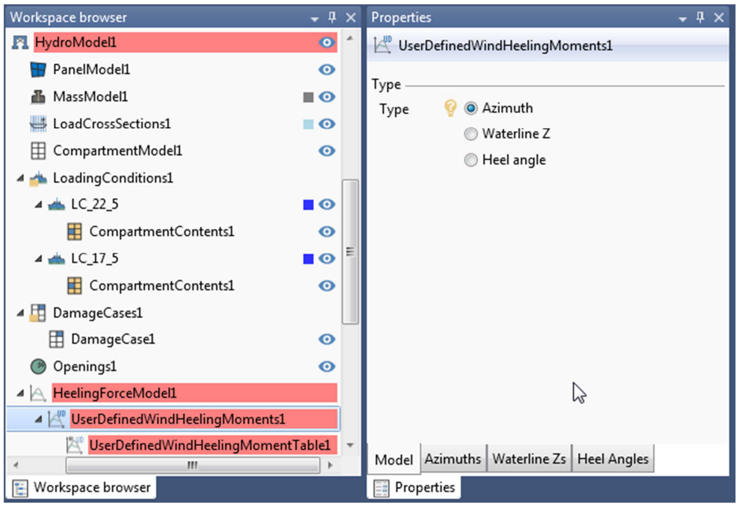
ElementModel2 is selected as the structure to hold wind and thrust force, and a translation of 7.5 m in z direction is applied.



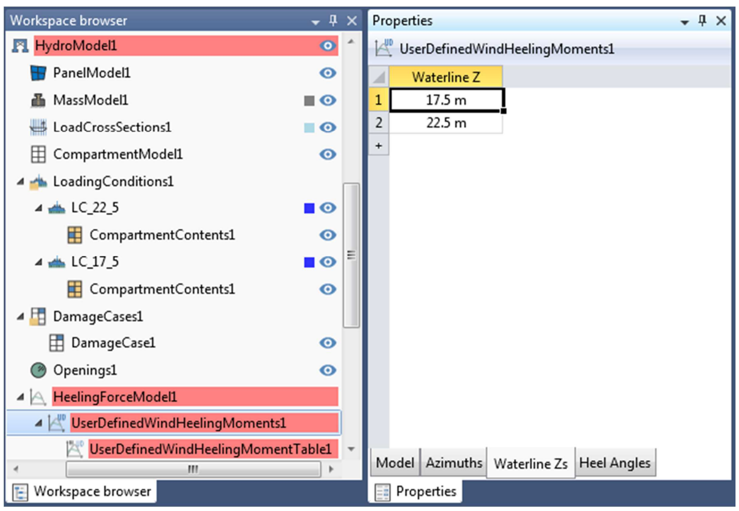
Step 20 – Create UserDefinedWindHeelingMoments

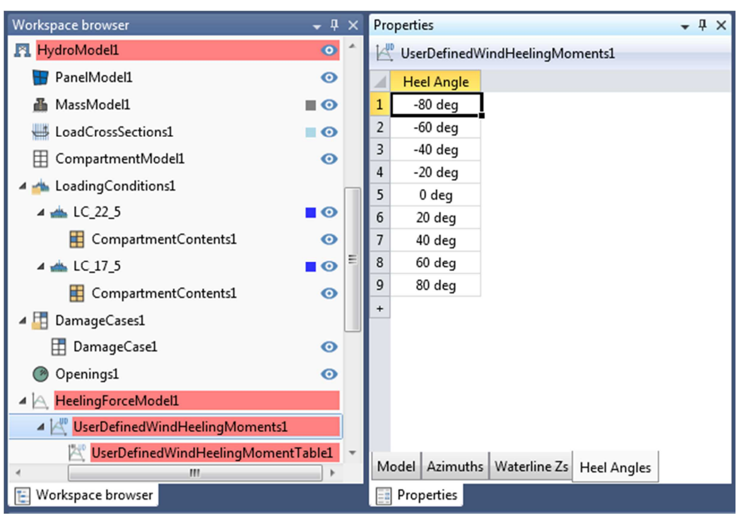


In this tutorial, the wind heeling moments are specified by the user which could be obtained from wind tunnel experiment. "Azimuth" type is used, which means each table is corresponding to an azimuth and contains the heeling moments for each waterline Z and heel angle.

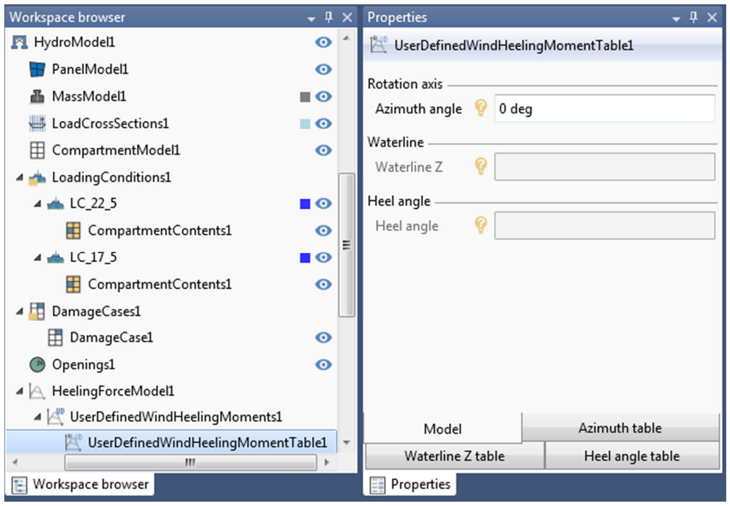


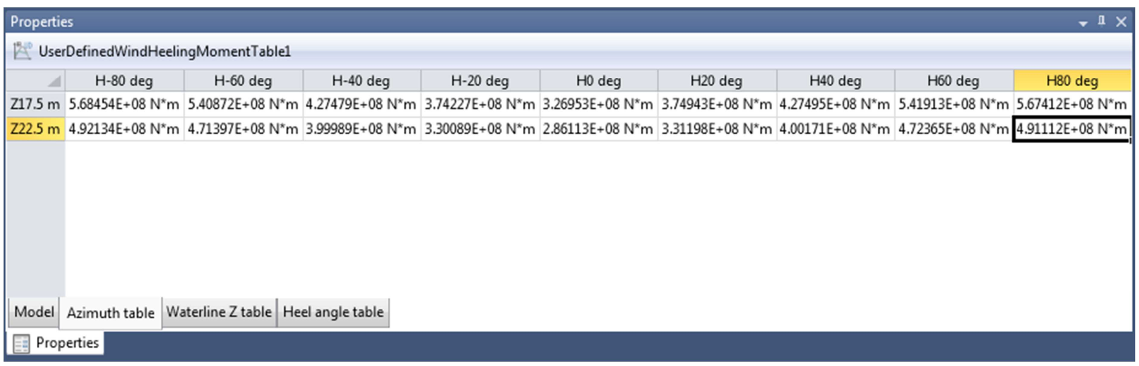
Define 2 waterlines and 9 heel angles.





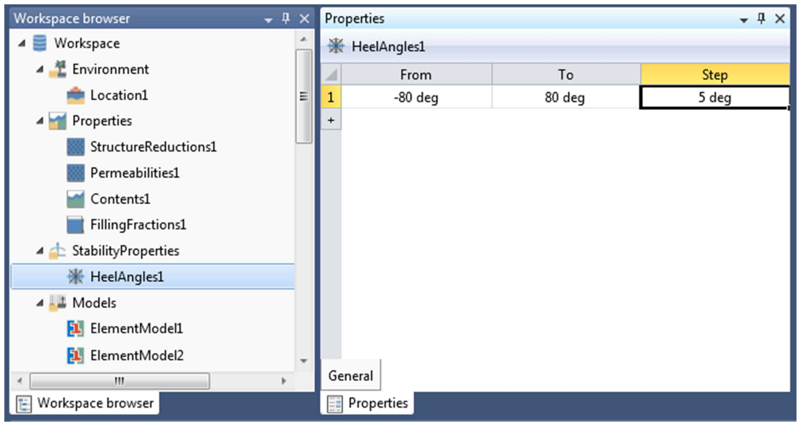
In this tutorial, we only have one wind heeling moment table for azimuth = 0 deg, and the values are shown as below.



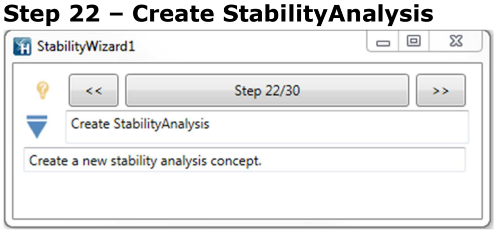


Step 21 – Create HeelAngles

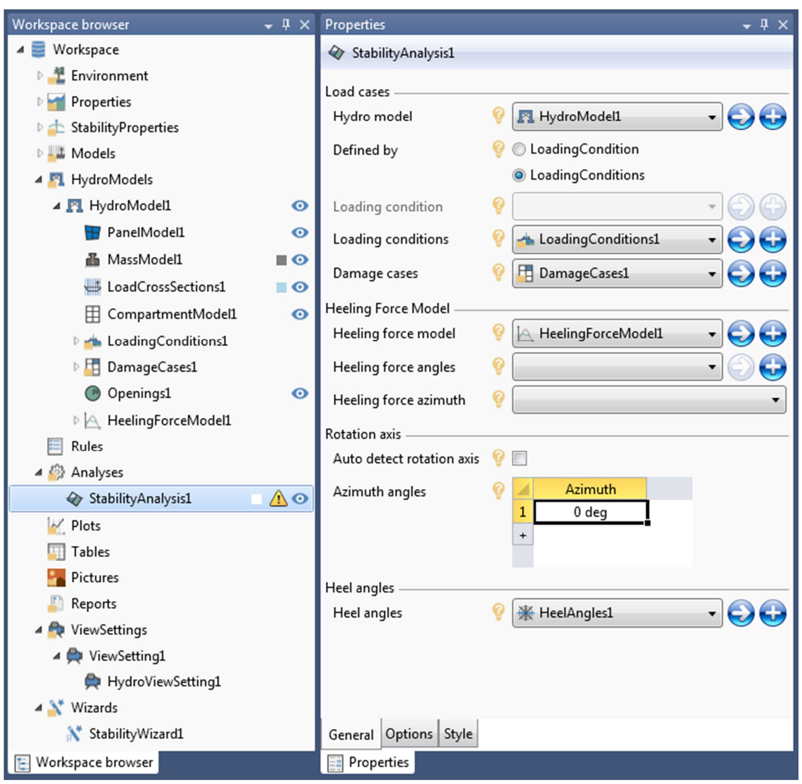
HeelAngles is used for GZ curve calculation, as well the wind heeling moments calculated by HydroD. In this tutorial we define the heel angles from -80 deg to 80deg with a 5 deg step.



Step 22 – Create StabilityAnalysis



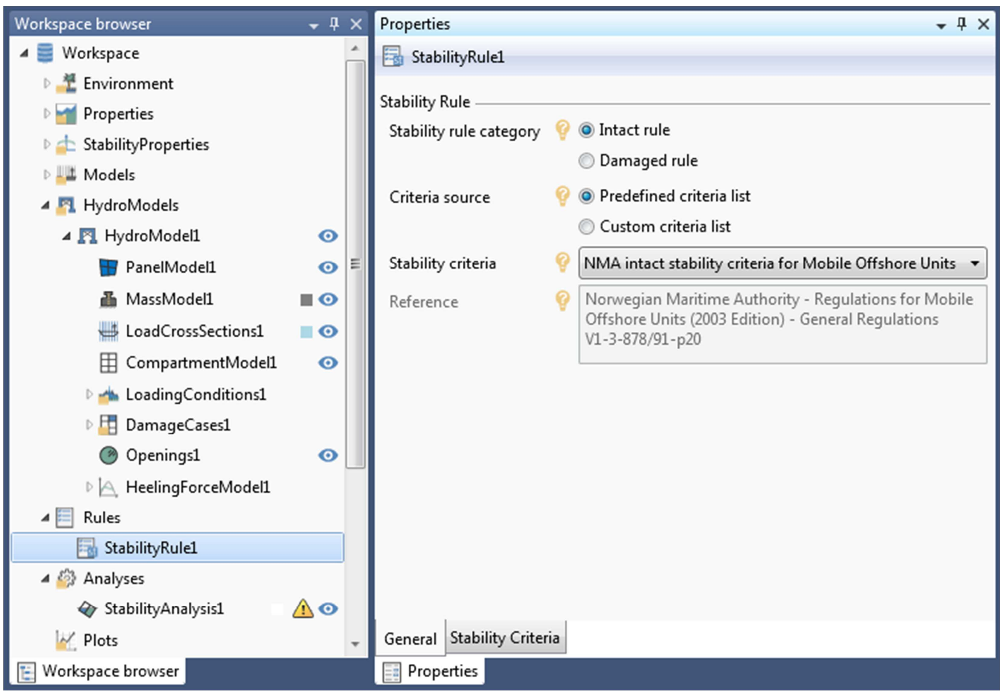
In this tutorial, we define the *StabilityAnalysis* as below.



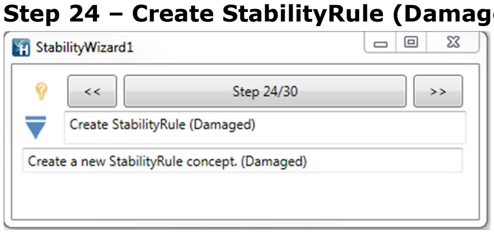
Step 23 – Create StabilityRule (Intact)

In this step, a *StabilityRule* is defined for the intact stability rule check.

We use NMA intact stability criteria in the tutorial.

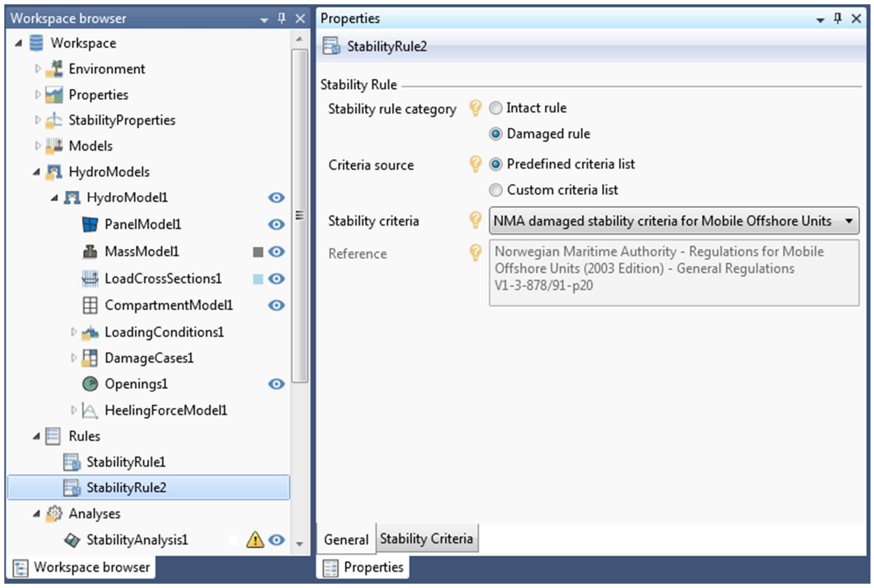


Step 24 – Create StabilityRule (Damaged)

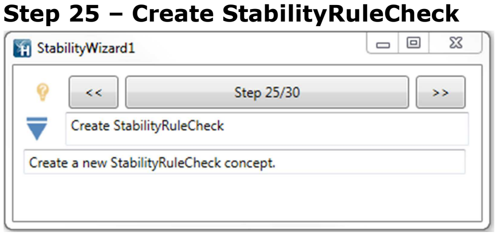


In this step, a *StabilityRule* is defined for the damaged stability rule check.

We use NMA damaged stability criteria in the tutorial.

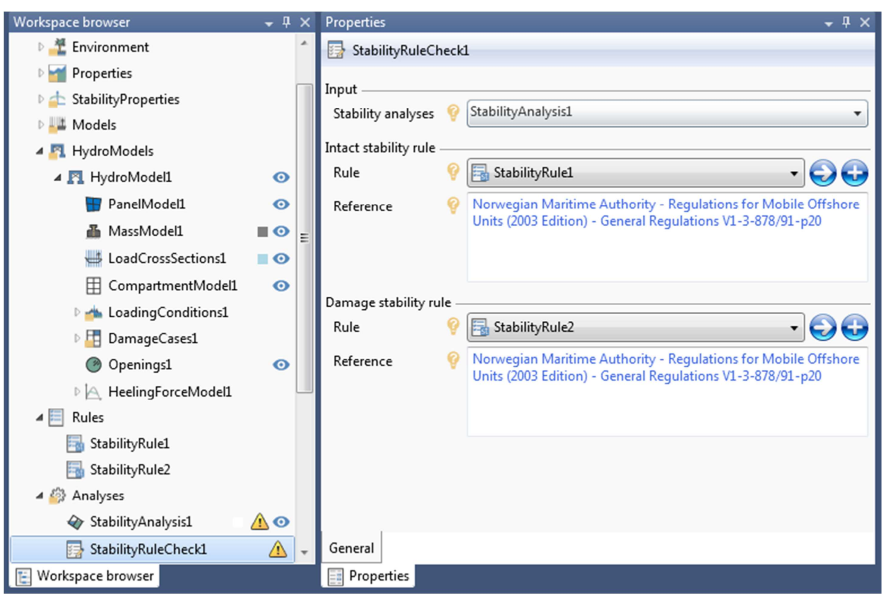


Step 25 – Create StabilityRuleCheck

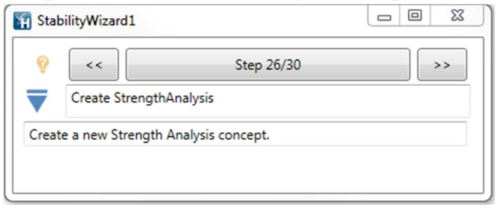


StabilityRuleCheck refers to one or several StabilityAnalysis. It will do the rule check with respect to the selected rules for each loading condition, after the StabilityAnalysis is executed successfully.

In this tutorial, the *StabilityRuleCheck* is defined as below.

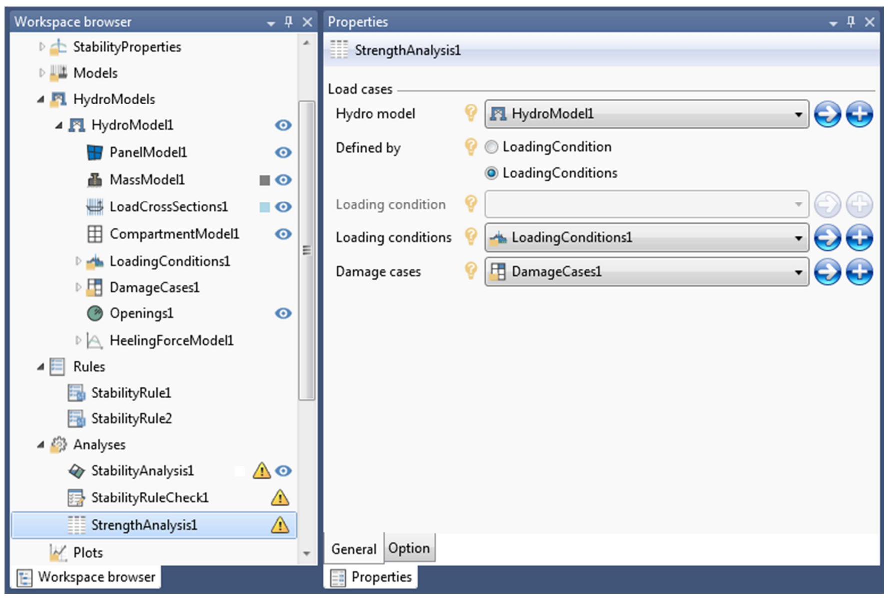


Step 26 – Create StrengthAnalysis

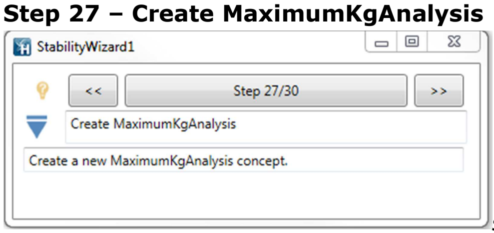


StrengthAnalysis is for the calculation of forces and moments on cross sections in still water.

Similar to *StabilityAnalysis*, we define the *StrengthAnalysis* by *LoadingConditions1* combined with *DamageCases1*.



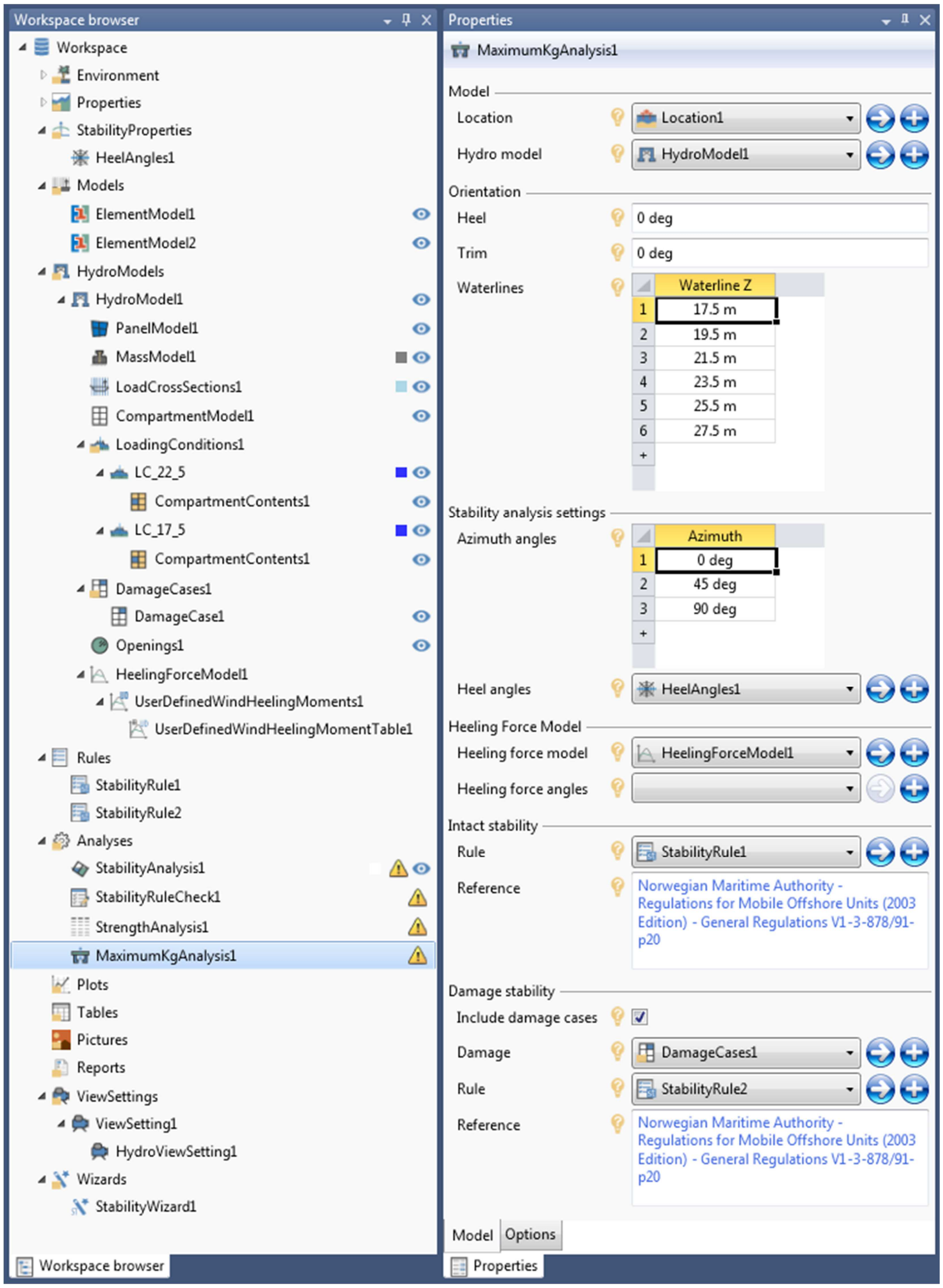
Step 27 – Create MaximumKgAnalysis



s

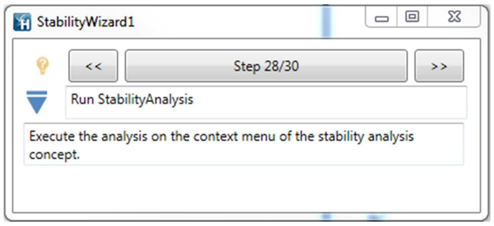
MaximumKgAnalysis is to calculate the maximum KG for a range of waterlines and azimuths with respect to intact and damage stability rule.

In this tutorial, a *MaximumKgAnalysis* is defined as below.



Workspace browser
Workspace
Environment
Properties
StabilityProperties
HeelAngles1
Models
ElementModel1
ElementModel2
HydroModels
HydroModel1
PanelModel1
MassModel1
LoadCrossSections1
CompartmentModel1
LoadingConditions1
LC_22_5
CompartmentContents1
LC_17_5
CompartmentContents1
DamageCases1
DamageCase1
Openings1
HeelingForceModel1
UserDefinedWindHeelingMoments1
UserDefinedWindHeelingMomentTable1
Rules
StabilityRule1
StabilityRule2
Analyses
StabilityAnalysis1
StabilityRuleCheck1
StrengthAnalysis1
MaximumKgAnalysis1
Plots
Tables
Pictures
Reports
ViewSettings
ViewSetting1
HydroViewSetting1
Wizards
StabilityWizard1
Workspace browser
Properties
MaximumKgAnalysis1
Model
Location
Location1
Hydro model
HydroModel1
Orientation
Heel
0 deg
Trim
0 deg
Waterlines
Waterline Z
1 17.5 m
2 19.5 m
3 21.5 m
4 23.5 m
5 25.5 m
6 27.5 m
Stability analysis settings
Azimuth angles
Azimuth
1 0 deg
2 45 deg
3 90 deg
Heel angles
HeelAngles1
Heeling Force Model
Heeling force model
HeelingForceModel1
Heeling force angles
Intact stability
Rule
StabilityRule1
Reference
Norwegian Maritime Authority - Regulations for Mobile Offshore Units (2003 Edition) - General Regulations V1-3-878/91-p20
Damage stability
Include damage cases
Damage
DamageCases1
Rule
StabilityRule2
Reference
Norwegian Maritime Authority - Regulations for Mobile Offshore Units (2003 Edition) - General Regulations V1-3-878/91-p20
Model
Options
Properties

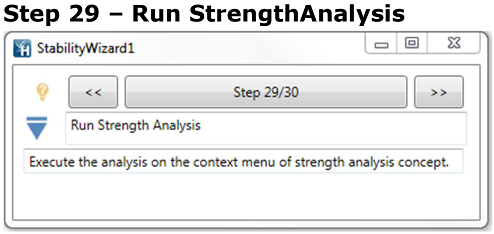
Step 28 – Run StabilityAnalysis



Click the *Step* button, then *StabilityAnalysis1* will run and the progress information is shown in the *Activity monitor* window.

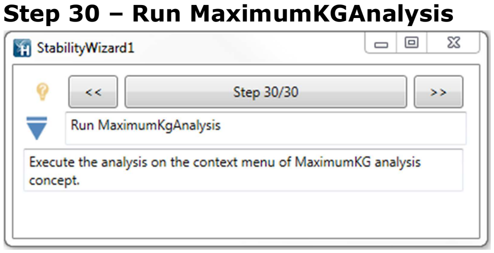
StabilityRuleCheck1, which refers to *StabilityAnalysis1*, will run automatically after *StabilityAnalysis1* is completed.

Step 29 – Run StrengthAnalysis



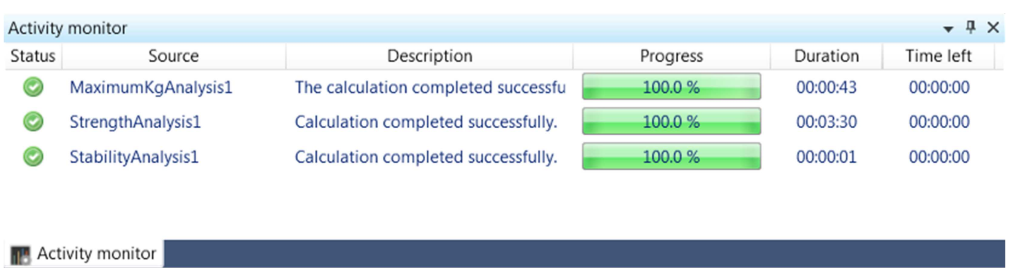
Click the *Step* button to run *StrengthAnalysis*.

Step 30 – Run MaximumKGAnalysis



Click the Step button to run *MaximumKgAnalysis*.

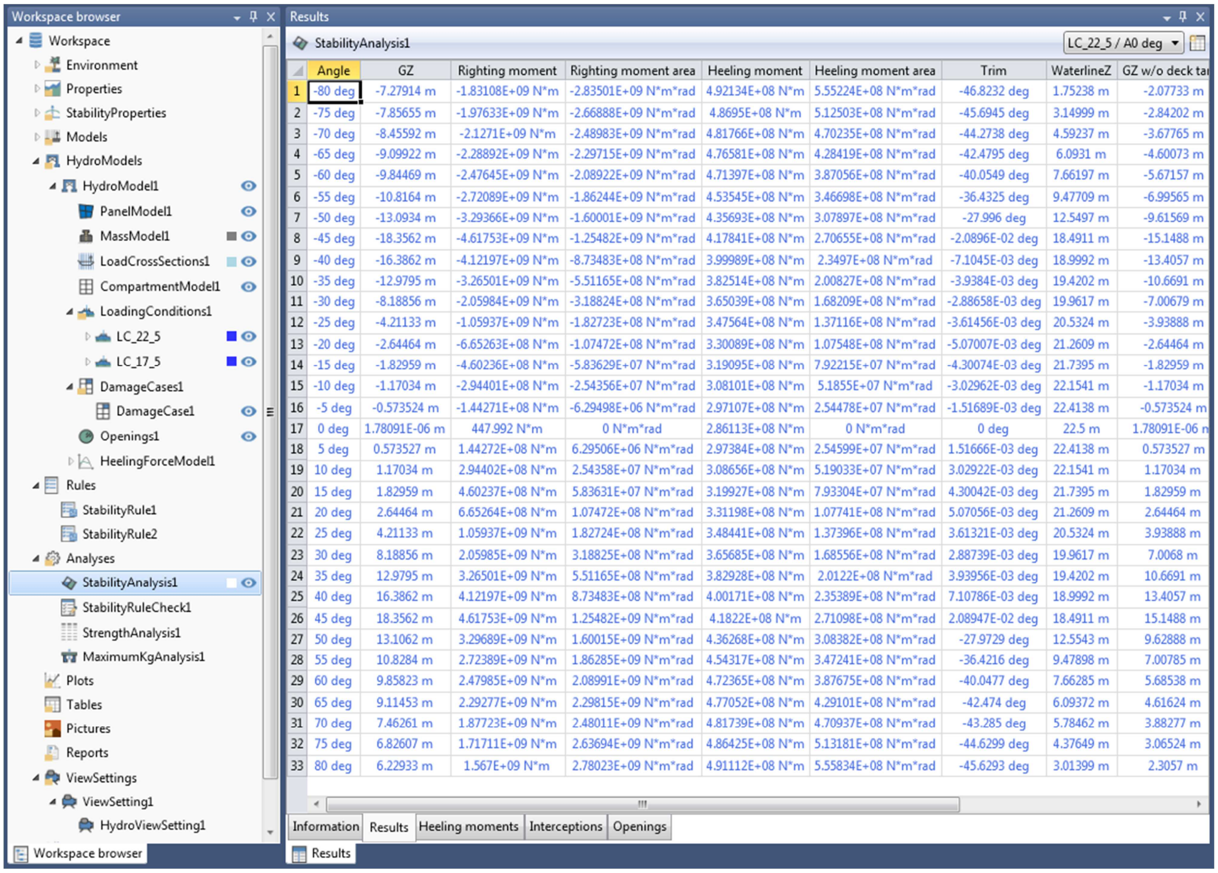
After all the analyses are completed, the Activity monitor would look as below.



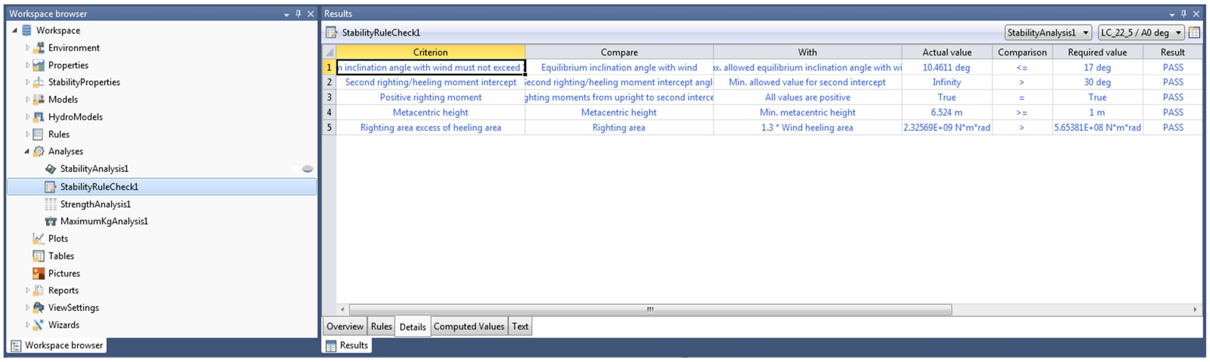
4 VIEW RESULTS

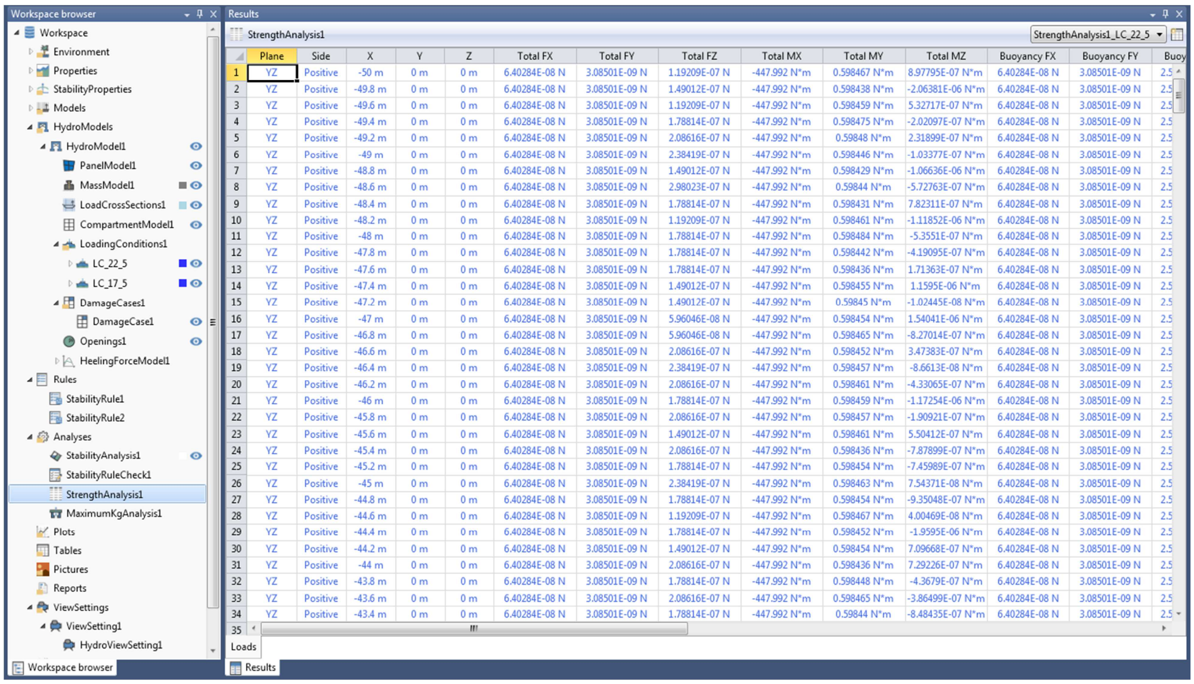
As all the analyses are completed, the results could be viewed in the *Results* window of corresponding analysis.

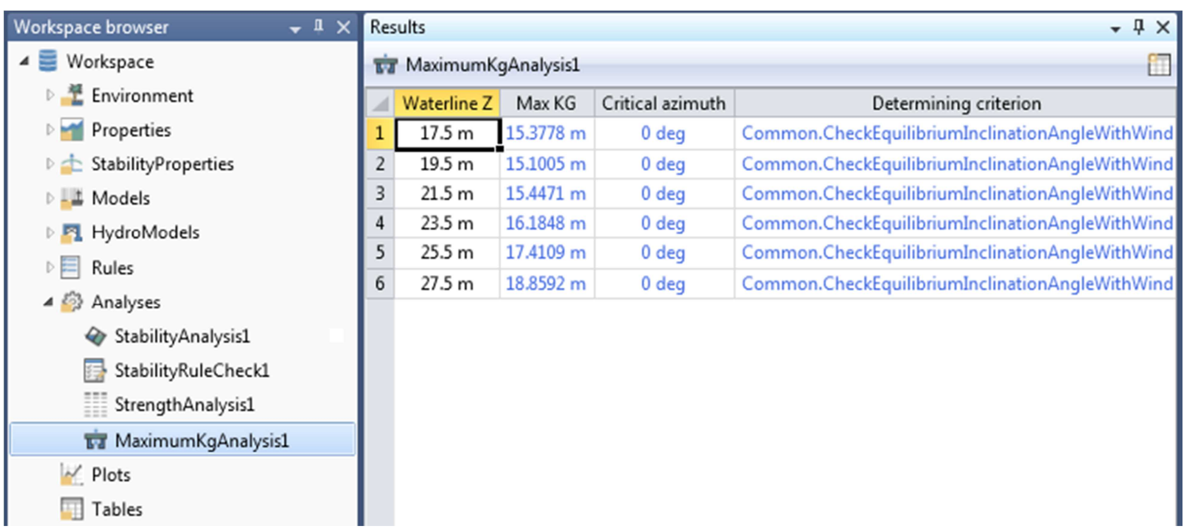
For *StabilityAnalysis*, *StabilityRuleCheck* and *StrengthAnalysis*, the results of each *LoadingCondition* could be shown by selecting the corresponding *LoadingCondition* from the dropdown list on the top right of *Results* window.



	Angle	GZ	Righting moment	Righting moment area	Heeling moment	Heeling moment area	Trim	WaterlineZ	GZ w/o deck ta
1	-80 deg	-7.27914 m	-1.83108E+09 N*m	-2.83501E+09 N*m*rad	4.92134E+08 N*m	5.55224E+08 N*m*rad	-46.8232 deg	1.75238 m	-2.07733 m
2	-75 deg	-7.85655 m	-1.97633E+09 N*m	-2.66888E+09 N*m*rad	4.8695E+08 N*m	5.12503E+08 N*m*rad	-45.6945 deg	3.14999 m	-2.84202 m
3	-70 deg	-8.45592 m	-2.1271E+09 N*m	-2.48983E+09 N*m*rad	4.81766E+08 N*m	4.70235E+08 N*m*rad	-44.2738 deg	4.59237 m	-3.67765 m
4	-65 deg	-9.09922 m	-2.28892E+09 N*m	-2.29715E+09 N*m*rad	4.76581E+08 N*m	4.28419E+08 N*m*rad	-42.4795 deg	6.0931 m	-4.60073 m
5	-60 deg	-9.84469 m	-2.47645E+09 N*m	-2.08922E+09 N*m*rad	4.71397E+08 N*m	3.87056E+08 N*m*rad	-40.0549 deg	7.66197 m	-5.67157 m
6	-55 deg	-10.8164 m	-2.72089E+09 N*m	-1.86244E+09 N*m*rad	4.53545E+08 N*m	3.46698E+08 N*m*rad	-36.4325 deg	9.47709 m	-6.99565 m
7	-50 deg	-13.0934 m	-3.29366E+09 N*m	-1.60001E+09 N*m*rad	4.35693E+08 N*m	3.07897E+08 N*m*rad	-27.996 deg	12.5497 m	-9.61569 m
8	-45 deg	-18.3562 m	-4.61753E+09 N*m	-1.25482E+09 N*m*rad	4.17841E+08 N*m	2.70655E+08 N*m*rad	-2.0896E-02 deg	18.4911 m	-15.1488 m
9	-40 deg	-16.3862 m	-4.12197E+09 N*m	-8.73483E+08 N*m*rad	3.99989E+08 N*m	2.3497E+08 N*m*rad	-7.1045E-03 deg	18.9992 m	-13.4057 m
10	-35 deg	-12.9795 m	-3.26501E+09 N*m	-5.51165E+08 N*m*rad	3.82514E+08 N*m	2.00827E+08 N*m*rad	-3.9384E-03 deg	19.4202 m	-10.6691 m
11	-30 deg	-8.18856 m	-2.05984E+09 N*m	-3.18824E+08 N*m*rad	3.65039E+08 N*m	1.68209E+08 N*m*rad	-2.88658E-03 deg	19.9617 m	-7.00679 m
12	-25 deg	-4.21133 m	-1.05937E+09 N*m	-1.82723E+08 N*m*rad	3.47564E+08 N*m	1.37116E+08 N*m*rad	-3.61456E-03 deg	20.5324 m	-3.93888 m
13	-20 deg	-2.64464 m	-6.65263E+08 N*m	-1.07472E+08 N*m*rad	3.30089E+08 N*m	1.07548E+08 N*m*rad	-5.07007E-03 deg	21.2609 m	-2.64464 m
14	-15 deg	-1.82959 m	-4.60236E+08 N*m	-5.83629E+07 N*m*rad	3.19095E+08 N*m	7.92215E+07 N*m*rad	-4.30074E-03 deg	21.7395 m	-1.82959 m
15	-10 deg	-1.17034 m	-2.94401E+08 N*m	-2.54356E+07 N*m*rad	3.08101E+08 N*m	5.1855E+07 N*m*rad	-3.02962E-03 deg	22.1541 m	-1.17034 m
16	-5 deg	-0.573524 m	-1.44271E+08 N*m	-6.29498E+06 N*m*rad	2.97107E+08 N*m	2.54478E+07 N*m*rad	-1.51689E-03 deg	22.4138 m	-0.573524 m
17	0 deg	1.78091E-06 m	447.992 N*m	0 N*m*rad	2.86113E+08 N*m	0 N*m*rad	0 deg	22.5 m	1.78091E-06 n
18	5 deg	0.573527 m	1.44272E+08 N*m	6.29506E+06 N*m*rad	2.97384E+08 N*m	2.54599E+07 N*m*rad	1.51666E-03 deg	22.4138 m	0.573527 m
19	10 deg	1.17034 m	2.94402E+08 N*m	2.54358E+07 N*m*rad	3.08656E+08 N*m	5.19033E+07 N*m*rad	3.02922E-03 deg	22.1541 m	1.17034 m
20	15 deg	1.82959 m	4.60237E+08 N*m	5.83631E+07 N*m*rad	3.19927E+08 N*m	7.93304E+07 N*m*rad	4.30042E-03 deg	21.7395 m	1.82959 m
21	20 deg	2.64464 m	6.65264E+08 N*m	1.07472E+08 N*m*rad	3.31198E+08 N*m	1.07741E+08 N*m*rad	5.07056E-03 deg	21.2609 m	2.64464 m
22	25 deg	4.21133 m	1.05937E+09 N*m	1.82724E+08 N*m*rad	3.48441E+08 N*m	1.37396E+08 N*m*rad	3.61321E-03 deg	20.5324 m	3.93888 m
23	30 deg	8.18856 m	2.05985E+09 N*m	3.18825E+08 N*m*rad	3.65685E+08 N*m	1.68556E+08 N*m*rad	2.88739E-03 deg	19.9617 m	7.0068 m
24	35 deg	12.9795 m	3.26501E+09 N*m	5.51165E+08 N*m*rad	3.82928E+08 N*m	2.0122E+08 N*m*rad	3.93956E-03 deg	19.4202 m	10.6691 m
25	40 deg	16.3862 m	4.12197E+09 N*m	8.73483E+08 N*m*rad	4.00171E+08 N*m	2.35389E+08 N*m*rad	7.10786E-03 deg	18.9992 m	13.4057 m
26	45 deg	18.3562 m	4.61753E+09 N*m	1.25482E+09 N*m*rad	4.1822E+08 N*m	2.71098E+08 N*m*rad	2.08947E-02 deg	18.4911 m	15.1488 m
27	50 deg	13.1062 m	3.29689E+09 N*m	1.60015E+09 N*m*rad	4.36268E+08 N*m	3.08382E+08 N*m*rad	-27.9729 deg	12.5543 m	9.62888 m
28	55 deg	10.8284 m	2.72389E+09 N*m	1.86285E+09 N*m*rad	4.54317E+08 N*m	3.47241E+08 N*m*rad	-36.4216 deg	9.47898 m	7.00785 m
29	60 deg	9.85823 m	2.47985E+09 N*m	2.08991E+09 N*m*rad	4.72365E+08 N*m	3.87675E+08 N*m*rad	-40.0477 deg	7.66285 m	5.68538 m
30	65 deg	9.11453 m	2.29277E+09 N*m	2.29815E+09 N*m*rad	4.77052E+08 N*m	4.29101E+08 N*m*rad	-42.474 deg	6.09372 m	4.61624 m
31	70 deg	7.46261 m	1.87723E+09 N*m	2.48011E+09 N*m*rad	4.81739E+08 N*m	4.70937E+08 N*m*rad	-43.285 deg	5.78462 m	3.88277 m
32	75 deg	6.82607 m	1.71711E+09 N*m	2.63694E+09 N*m*rad	4.86425E+08 N*m	5.13181E+08 N*m*rad	-44.6299 deg	4.37649 m	3.06524 m
33	80 deg	6.22933 m	1.567E+09 N*m	2.78023E+09 N*m*rad	4.91112E+08 N*m	5.55834E+08 N*m*rad	-45.6293 deg	3.01399 m	2.3057 m

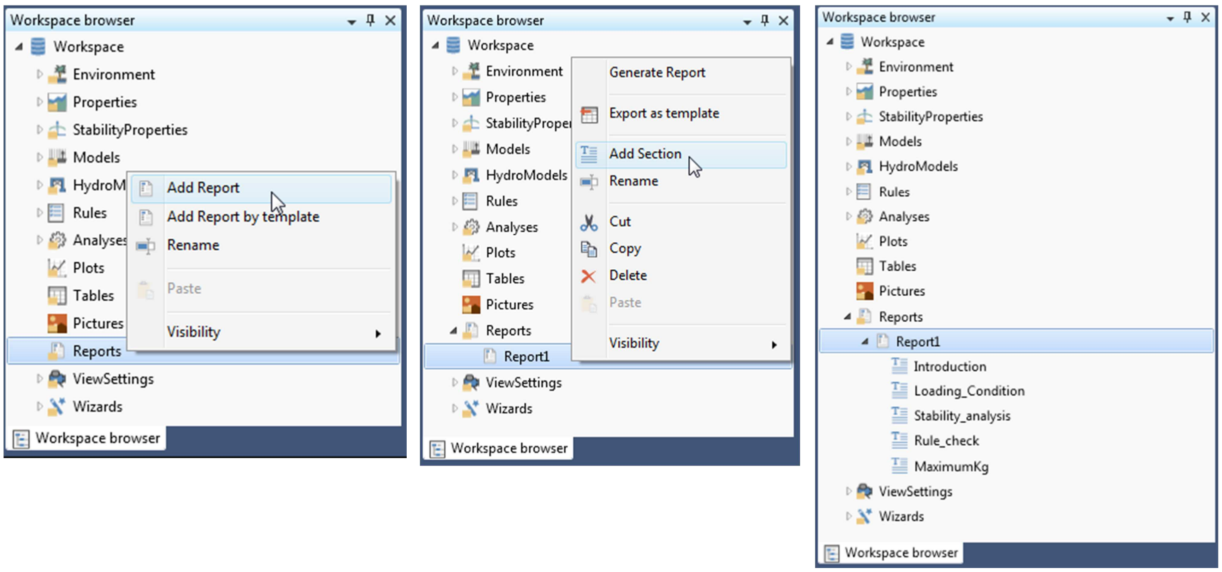




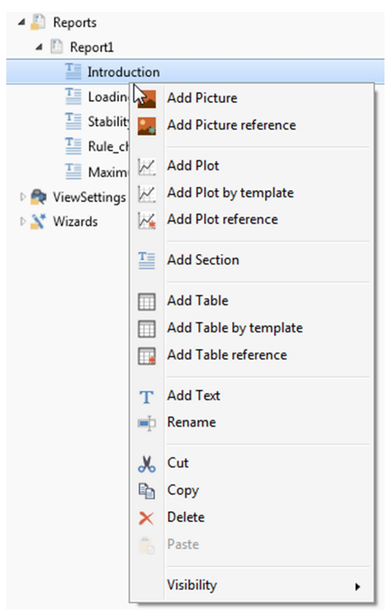

Workspace browser
Workspace
Environment
Properties
StabilityProperties
Models
HydroModels
Rules
Analyses
StabilityAnalysis1
StabilityRuleCheck1
StrengthAnalysis1
MaximumKgAnalysis1
Plots
Tables
Results
MaximumKgAnalysis1
Waterline Z
Max KG
Critical azimuth
Determining criterion
1 17.5 m 15.3778 m 0 deg Common.CheckEquilibriumInclinationAngleWithWind
2 19.5 m 15.1005 m 0 deg Common.CheckEquilibriumInclinationAngleWithWind
3 21.5 m 15.4471 m 0 deg Common.CheckEquilibriumInclinationAngleWithWind
4 23.5 m 16.1848 m 0 deg Common.CheckEquilibriumInclinationAngleWithWind
5 25.5 m 17.4109 m 0 deg Common.CheckEquilibriumInclinationAngleWithWind
6 27.5 m 18.8592 m 0 deg Common.CheckEquilibriumInclinationAngleWithWind

5 CREATE REPORT

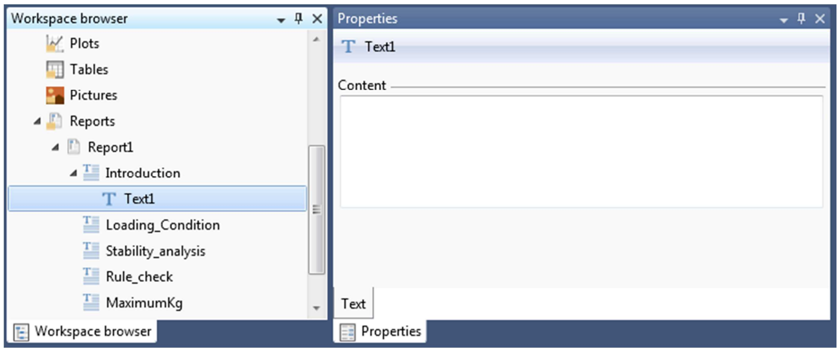
A *Report* can be added form the context menu of *Reports* folder in the *Workspace browser*. And several sections can be added to a report. In this tutorial, add five sections as below.



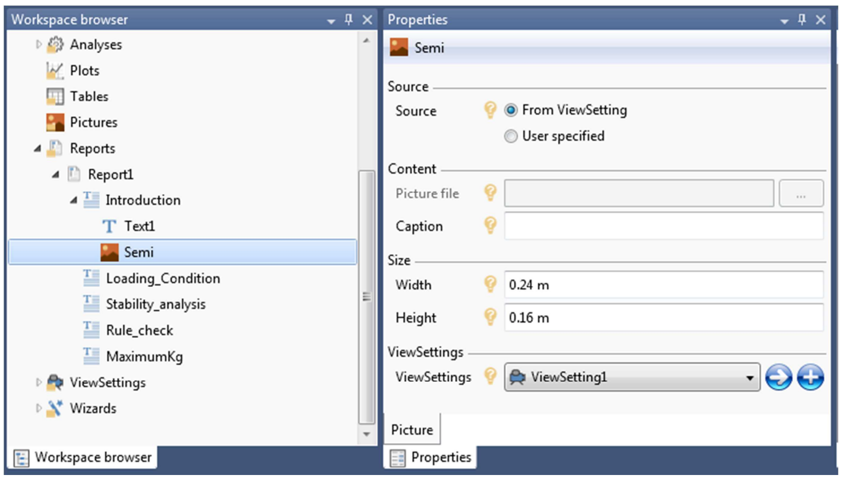
You can add text, picture, plot or table to a *Section*.



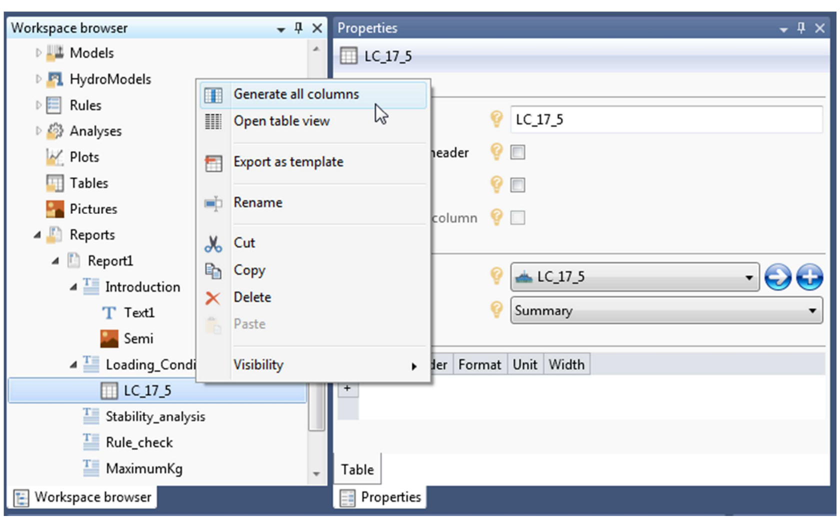
Text could be edited in the *Properties* window and be placed in the corresponding place in the final report.



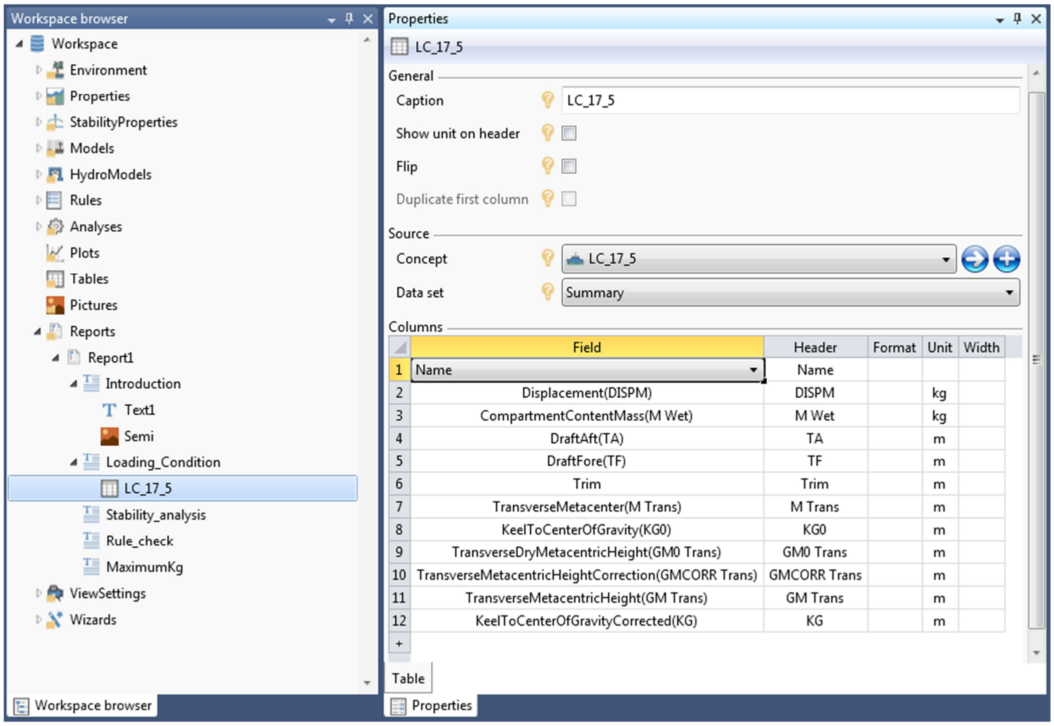
A *picture* could the screen-shot of the 3D window or an external image file.



A *Table* is added to the section of *Loading_Condition*, with the name *LC_17_5*, and has the settings as below. In the context menu, execute the command of *Generate all columns*. All the columns would be filled to the *Table*.

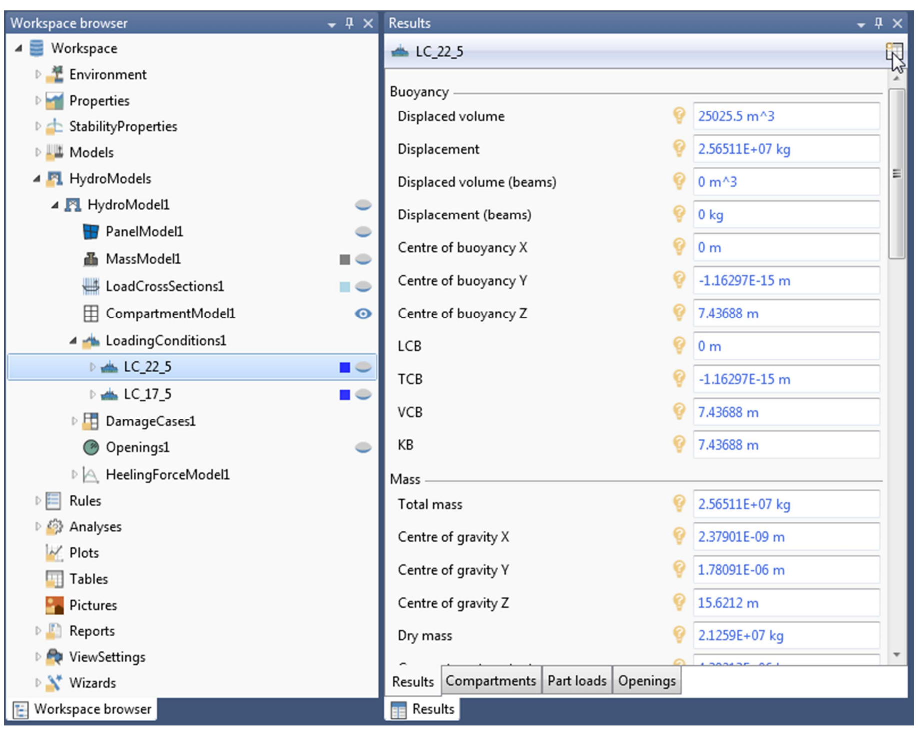


All the columns of the summary of *LC_17_5* are listed in the *Columns*, where you could modify further.

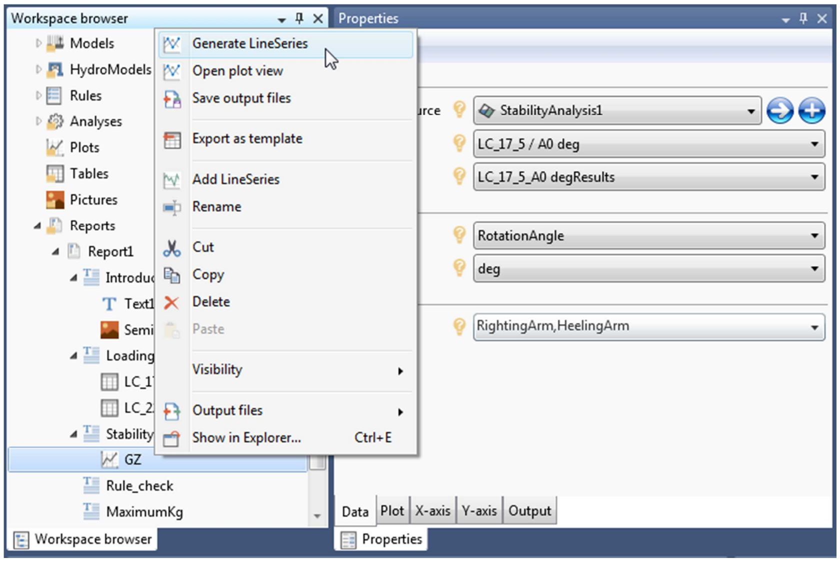


Double-click *LC_17_5* in *Workspace browser*, the *Table* window will open with the table just defined.

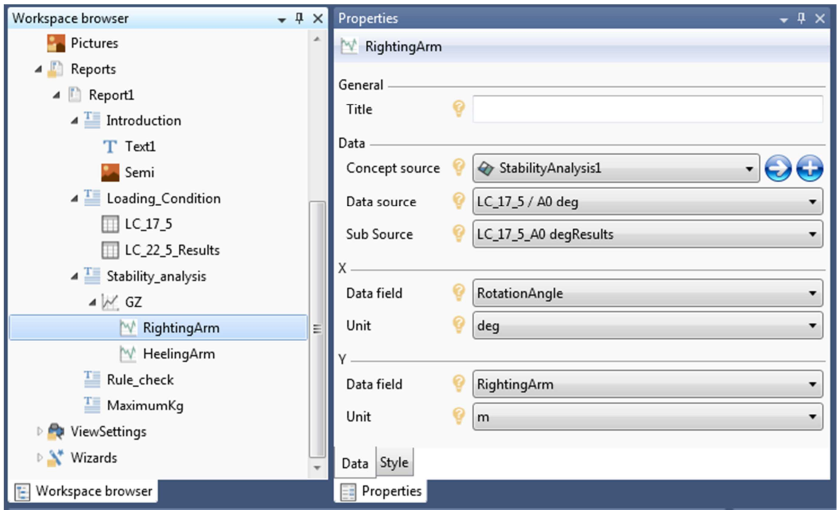
Another convenient way to add a *Table* is to use the *Generate table* button in the *Results* window. Switch to the *Results* window of *LC_22_5*, and click *Generate table* button at the top right corner. A *Table* will be generated in the *Tables* folder. Cut the Table and paste it to *Report1 -> Loading_Condition*.



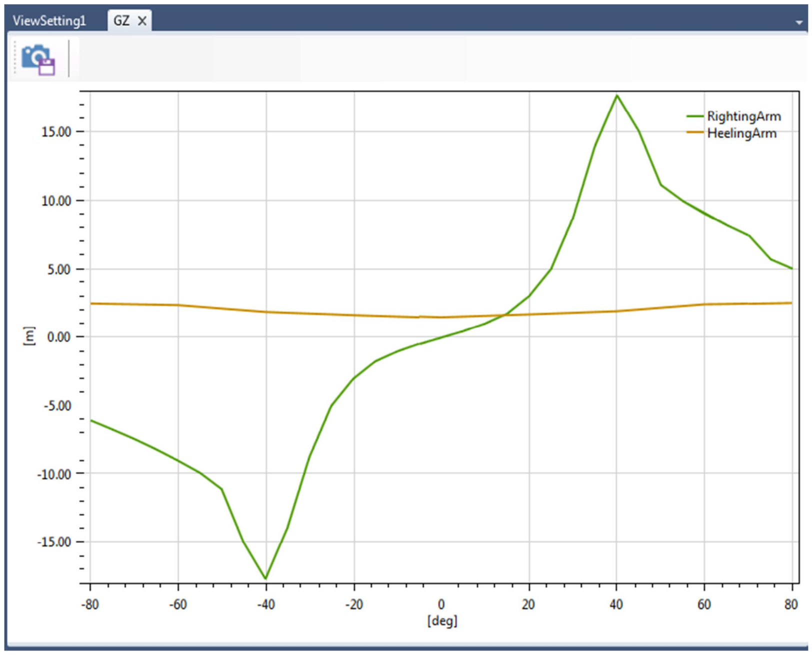
Next, we add a *Plot* to *Stability_analysis* section and rename it to *GZ*. Define the properties of the plot as below, and then execute the command *Generate LineSeries* in the context menu of the plot.



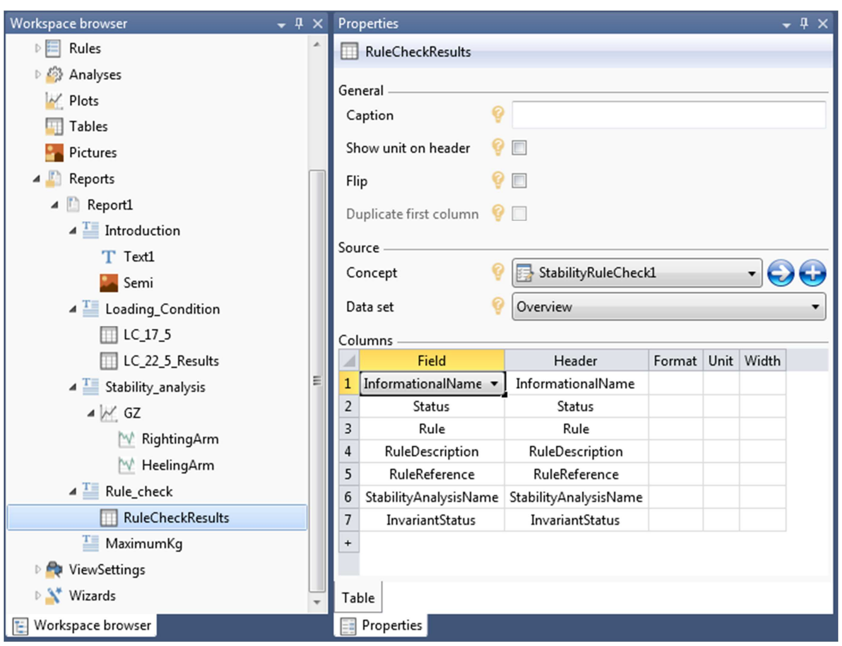
The lines of righting arm and heeling arm are added.



Double click *GZ*, the two lines will be shown in the *Plot* window.

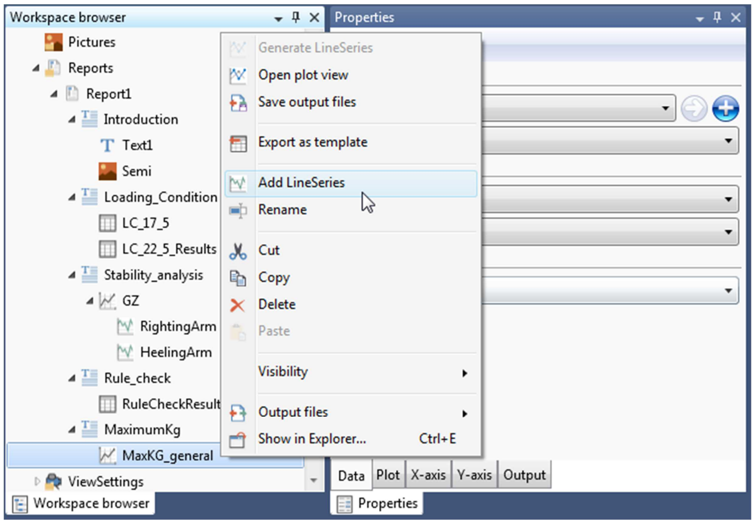


We add a *Table* to *Rule_check* section in a way similar to those in *Loading_Condition* section.

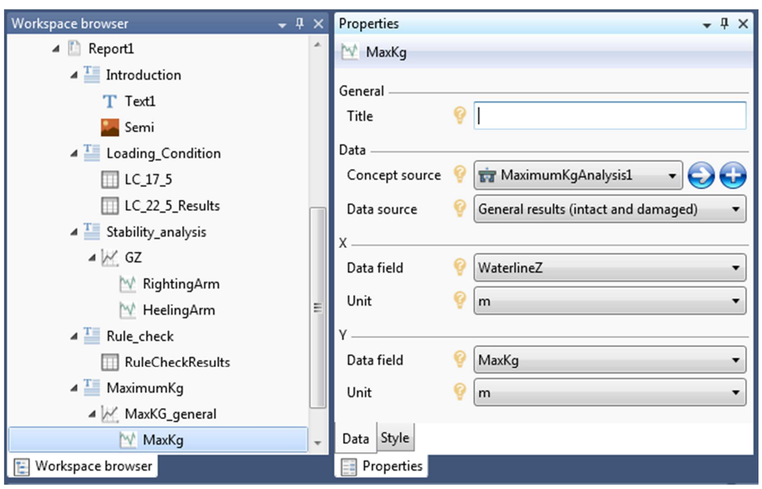


In the section of *MaximumKg*, a *Plot* is added to show the maximum KG curve.

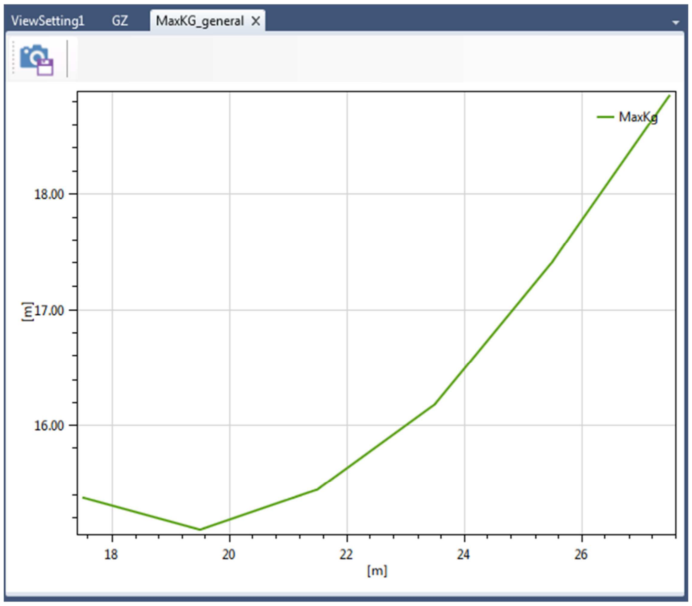
In the context menu, we add a *LineSeries* manually.



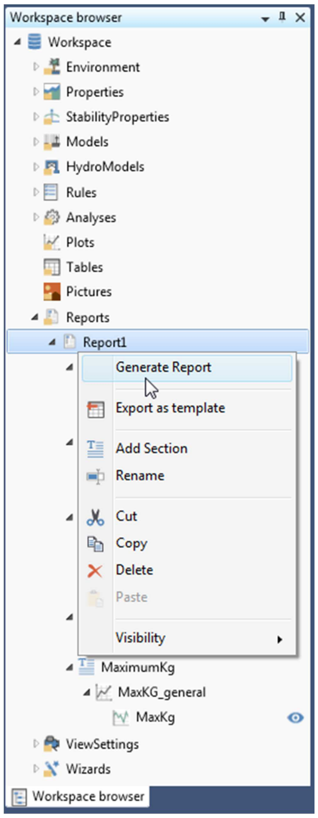
The settings in the *Properties* view are shown as below.



The curve can be shown in the *Plot* window.



As all the content added, a Word report could be generated by the command *Generate Report*.



About DNV GL

Driven by our purpose of safeguarding life, property and the environment, DNV GL enables organizations to advance the safety and sustainability of their business. We provide classification and technical assurance along with software and independent expert advisory services to the maritime, oil and gas, and energy industries. We also provide certification services to customers across a wide range of industries. Operating in more than 100 countries, our 16,000 professionals are dedicated to helping our customers make the world safer, smarter and greener.

Software

DNV GL is the world-leading provider of software for a safer, smarter and greener future in the energy, process and maritime industries. Our solutions support a variety of business critical activities including design and engineering, risk assessment, asset integrity and optimization, QHSE, and ship management. Our worldwide presence facilitates a strong customer focus and efficient sharing of industry best practice and standards.