SESAM EXAMPLE

Soil Linearization along the Pile

Sesam Example

Soil Linearization along the Pile

Date: June 2025

Prepared by: Digital Solutions at DNV

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

E-mail sales: digital@dnv.com

Table of contents

1 Introduction

The model used in the in-place analysis includes piles and soil (nonlinear soil curves). These must be replaced with linearized springs in order to perform dynamic or eigenvalue analysis with Sestra module.

The pile linearization procedures are implemented in Splice, and the derived linearized springs can be imported into model from GenE.

There are two ways to perform pile linearization, including

- Method A – replace piles and soil with linearized stiffness matrix at pilehead
- Method B – replace piles and soil with beams and linearized springs along the pile

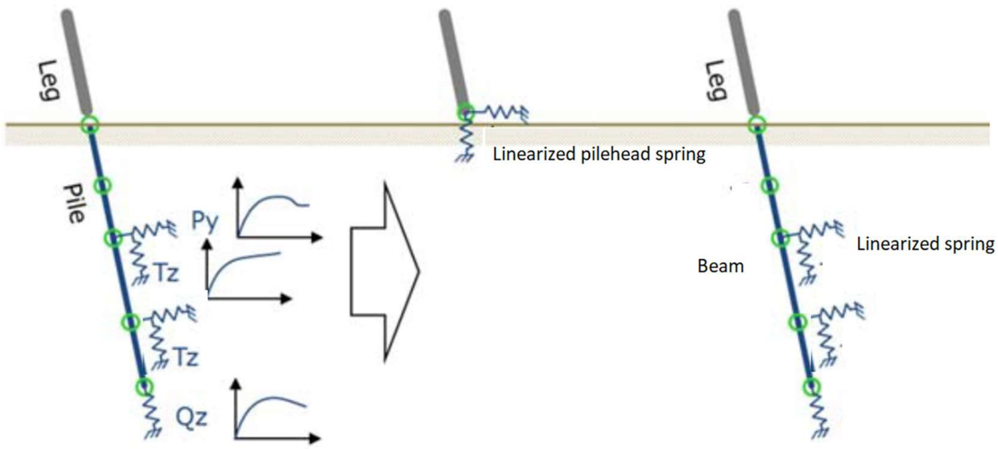


Figure 1.1 Replace pile and soil with linearized springs

Method A has been implemented in Splice for many years, but Method B is a new feature from Splice8.1 and GeniE8.8.

From Splice8.1, it can print soil springs at each of the pile node coordinates as well as point masses representing the mass of soil/fluid inside the pile. These springs and point masses can easily be imported into GeniE so that the user can do a linear analysis without the piles and nonlinear springs and instead include the piles as beams with the linear springs and pint masses. This is relevant for eigenvalue analysis, dynamic analysis, superelement generation for wind turbine tools and other types of linear analyses.

This example will focus on:

- How to derive the linearized springs along the pile
- How to import the linearized springs along the pile into the GeniE model
- How to execute linear analysis and eigenvalue analysis with the linearized springs along the pile
- Possible issues and limitations

The versions used in this example include Genie 8.12-02, Wajac7.13.00, Splice8.2-01, Sestra10.19-00.

DNV

2 Detailed Steps

Create a new GeniE workspace, set the default units to N and m. Import the gnx file **ExampleModel_Start.gnx**. This is a OWT jacket foundation, the soil curves, environmental conditions, load combinations and pile soil analysis have been included.

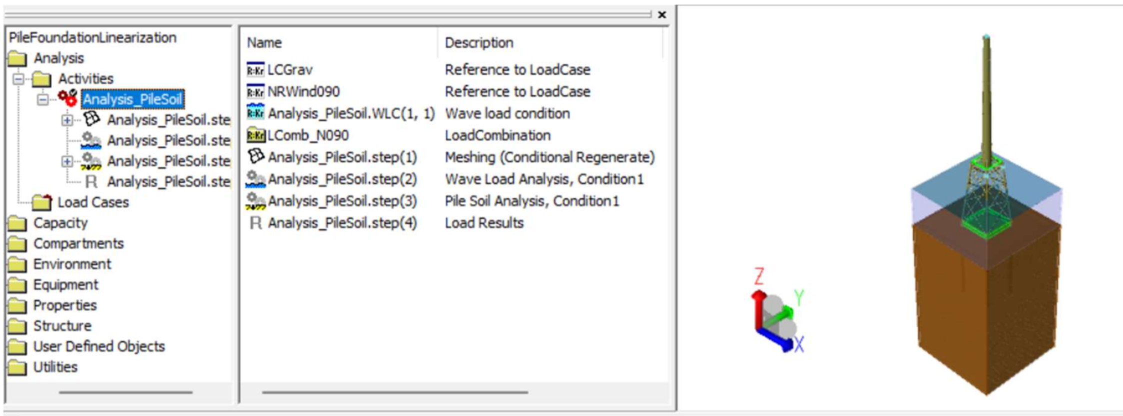


Figure 2.1 The imported model file

The pile characteristics with 1000kg/m3 density of soil/fluid inside pile has been assigned to the piles, which will be converted to point masses in the later section.

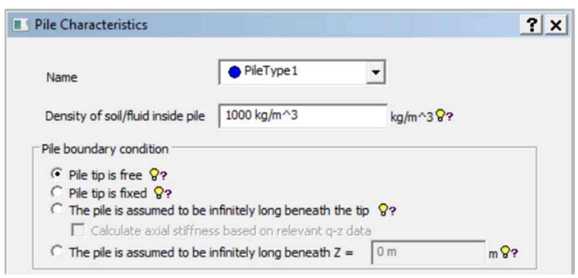


Figure 2.2 Pile characteristics

2.1 Execute Pile Soil Analysis

By default, the linearized springs will be derived for all the load cases solved by Sestra with this feature. In this example, the load combination LComb_N090 is selected to derive the linearized springs.

- Edit Pile Soil Analysis, go to *Sestra* > *Load Cases* to include LComb_N090 only.

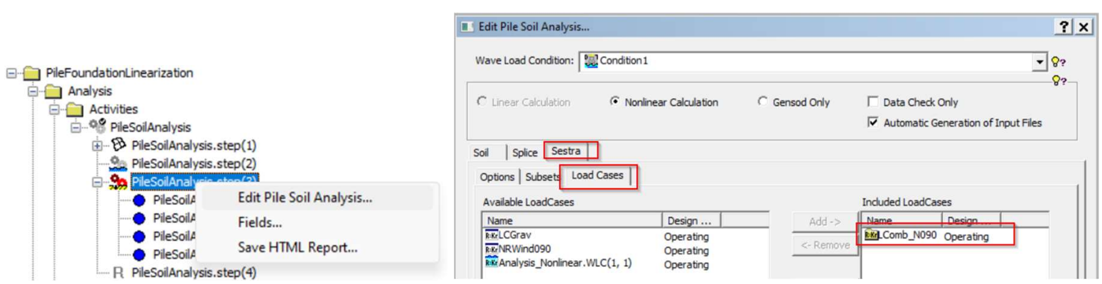


Figure 2.3 Define load cases used to derive linearized springs

- Run the analysis.

2.2 Check Results

After the analysis ran through, the files to the right will be stored in the analysis folder.

- Open **Splice.inp**, it shows that MISC(1) has been written as 3. With this value, a linear representation of the whole pile & soil will be printed on the file SPLICE_LINEARISEDPILE_T1.FEM.

```
************************* CONTROL SECTION
       1. CONFRC     OLD-FORCE-UNIT  = CONFRC * NEW-FORCE-UNIT  1MN  = 1000*1KN
       1. CONLTH     OLD-LENGTH-UNIT = CONLTH * NEW-LENGTH-UNIT  1M  = 3.28*1FT
        4 NPH        NUMBER OF PILE HEADS INCLUDING DUMMIES
        4 NLPH       NUMBER OF PILE HEADS WITH GIVEN LOADS
       11 JACK       CODE FOR PRESENCE OF JACKET (0=NO 1=YES)
        0 LOAPIL     CODE FOR PRESENCE OF LOADS FROM PILGEN (0=NO 1=YES)
       13 NUMVEC     NUMBER OF LOAD VECTORS TO BE ANALYZED
        0 ISTART     CODE IJ FOR SAVE/READ OF RE-START VALUES (I=SAVE J=READ)
        1 ISECM      CODE FOR SECOND ORDER MOMENTS (0=NEGLECT 1=INCLUDE)
        3 NURCHK     CROSS SECTION STRENGTH UR CHECK (0=NO 1=NORSOK 2=ISO-19902 3=API-WSD_2014)
     1111 IPRT1      PRINT CODE IJKL (L=PILES K=LOADS J=SOIL I=SOIL.DISP)
        1 JECHO      ECHO PRINT OF INPUT FILE NF5 TO FILE NF14 (0=NO 1=YES)
3  0  0  0  0            MISC(1) - MISC(5)  SPECIAL PURPOSE PARAMETERS
```

Figure 2.4 Related command in Splice.inp

Note: For pile soil analysis, the automatic generated Splice.inp file in GeniE8.8 and later versions will set MISC(1)=3 by default, this is a switch to turn on the feature pile linearization along the pile.

- Open **SPLICE_LINEARISEDPILE_T1.FEM** from the analysis folder. This file contains the pile node number, node coordinate, converted point masses, the linearized spring/stiffness matrix and its corresponding load case number. In the below example, a point mass of 397.62 and a spring of kx=ky=1.54022962E+07, kz=2.82912612E+08, and krz=8.47527659E+07 is defined at the point (11, 11, -82.5).

```
GNODE     1.64000000E+02  1.64000000E+02  6.00000000E+00  1.23456000E+05
GCOORD    1.64000000E+02  1.10000000E+01  1.10000000E+01 -8.25000000E+01   node number and its coordinate
BNMASS    1.64000000E+02  6.00000000E+00  3.97620010E+02  3.97620010E+02   point mass on pile node
          3.97620010E+02  0.00000000E+00  0.00000000E+00  0.00000000E+00
AMATRIX   1.20000000E+01  1.00000000E+00  0.00000000E+00  1.64000000E+02
          8.00000000E+00  0.00000000E+00  0.00000000E+00  0.00000000E+00   corresponding load case number
          1.00000000E+00  1.00000000E+00  1.00000000E+00  4.00000000E+00
AMDSTIFF  4.10000000E+01  1.00000000E+00  1.64000000E+02  1.64000000E+02
          6.00600000E+03  1.54022962E+07  0.00000000E+00  0.00000000E+00
          0.00000000E+00  0.00000000E+00  0.00000000E+00  0.00000000E+00
          1.54022962E+07  0.00000000E+00  0.00000000E+00  0.00000000E+00
          0.00000000E+00  0.00000000E+00  0.00000000E+00  2.82912612E+08   linear spring / stiffness matrix
          0.00000000E+00  0.00000000E+00  0.00000000E+00  0.00000000E+00
          0.00000000E+00  0.00000000E+00  0.00000000E+00  0.00000000E+00
          0.00000000E+00  0.00000000E+00  0.00000000E+00  0.00000000E+00
          0.00000000E+00  0.00000000E+00  0.00000000E+00  0.00000000E+00
          0.00000000E+00  0.00000000E+00  0.00000000E+00  0.00000000E+00
          8.47527659E+07
```

Figure 2.5 SPLICE_LINEARISEDPILE_T1.FEM

In the above FEM file, have no unit related information, the unit is determined by UNITS card in the model T1.FEM file.

```
UNITS     5.00000000E+00  1.00000000E+00  1.00000000E+00  1.00000000E+00
          1.00000000E+00
TDMATER   4.00000000E+00  1.00000000E+00  1.07000000E+02  0.00000000E+00
        Mat_JKT
```

Figure 2.6 The unit in T1.FEM

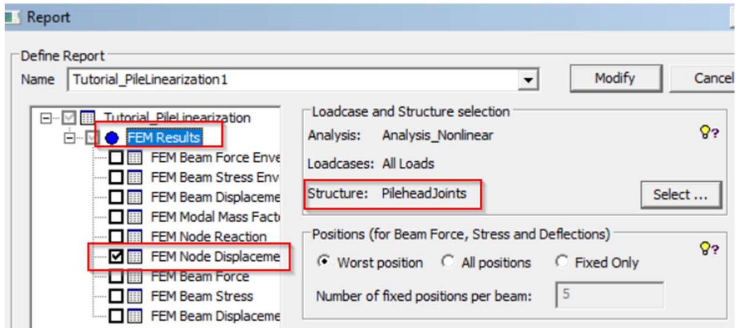
Note: All included load cases in the sestra solution are used to derive linearized springs and the springs are printed in the file SPLICE_LINEARISEDPILE_T1.FEM.

- Check load sum in Sestra.lis file.

```
Load sum for all result cases (internal number):
   result case;           tx;           ty;           tz;           rx;           ry;           rz
             1;  5.846145e+02;  2.803250e+06; -1.461431e+07; -2.040887e+08;  2.037481e+06; -4.020529e+05
```

Figure 2.7 Load Sum for derivation of soil linearized springs

- Go to *File* > *Save Report* > *FEM Results*, check **FEM Node Displacements** in set **PileheadJoints**.



Name	FEM Node	X [m]	Y [m]	Z [m]	DX [m]	DY [m]	DZ [m]	RX [deg]	RY [deg]	RZ [deg]	Complex
Jt31	71	-11	-11	-27.5	0.00260282	0.0207982	0.00194138	-0.112124	0.0108726	0.00185002	No
Jt32	73	11	-11	-27.5	-0.0028235	0.0204314	0.00186857	-0.110528	-0.0120043	-0.00232798	No
Jt33	83	-11	11	-27.5	-0.00583361	0.0243623	-0.00838627	-0.123699	-0.0230028	0.00115811	No
Jt34	93	11	11	-27.5	0.00612384	0.0241424	-0.00847065	-0.122618	0.0245164	-0.00152128	No

Figure 2.8 Pilehead nodal displacements derived from Save Report

The load sum and pilehead node displacements will be used in the comparison with linear analysis in section 2.6.

2.3 Import Linearized Springs along Pile Length

Go to *File* > *Import* > *Import linearised pile springs* to open the dialog window.

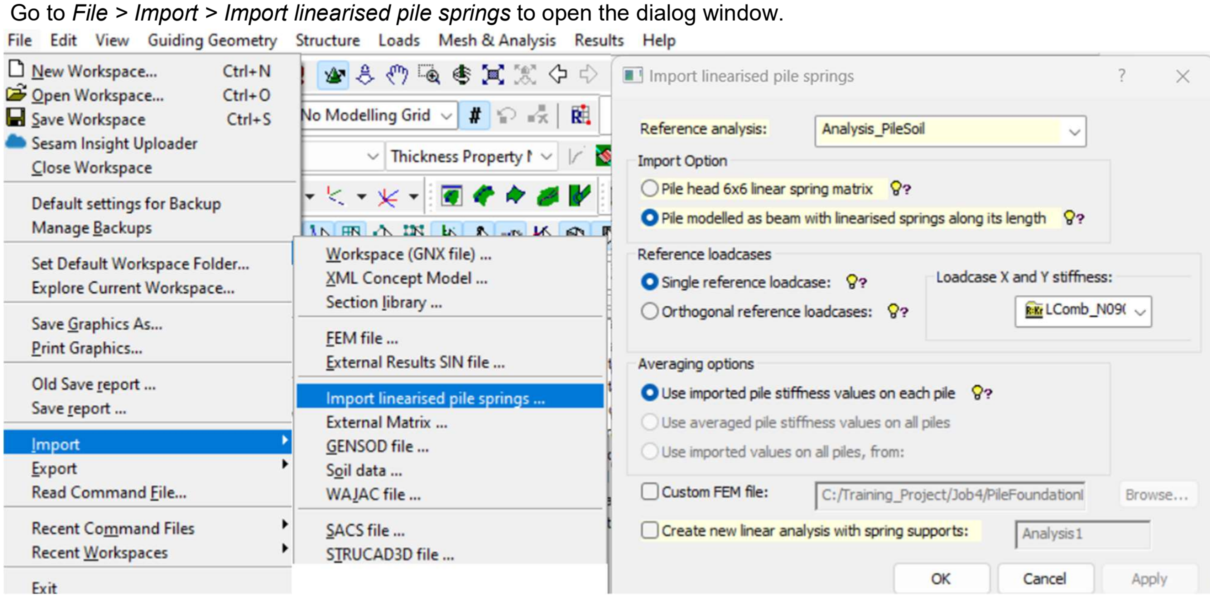


Figure 2.9 GUI for import linearized springs

- Select the refence analysis **Analysis_PileSoil** in which contains SPLICE_LINEARISEDPILE_T1.FEM
- Check the import option "**Pile modelled as beam with linearised springs along its length**"
- Select single reference load case and specify **Lcomb_N090**
- Uncheck the option "**Create new linear analysis with spring supports**", the new analysis will be created in the next section

Note: If check "Create new linear analysis with spring supports" and enter the analysis name, then a linear analysis like below will be generated automatically and the pile boundary condition under Edit mesh activity has been set as "Piles As Beams".

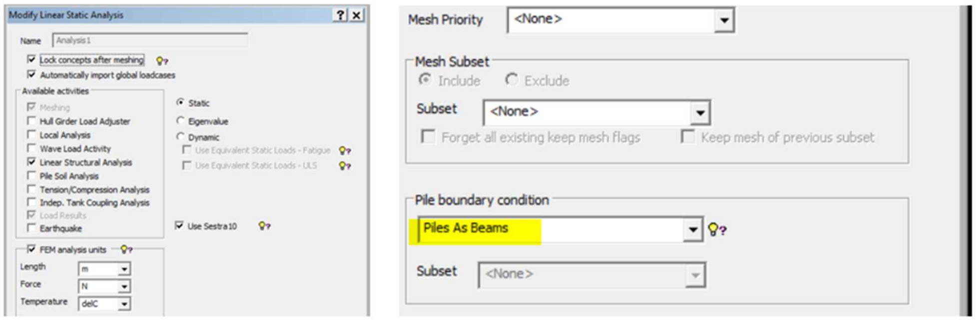


Figure 2.10 The setting when "create new linear analysis with spring supports" is checked

2.4 Check the Imported Linearized Springs and Point Masses

With the setting in the above Figure 2.9, click OK. The linearized springs derived from load case LComb_N090 and the point masses from pile characteristics has been imported successfully.

Check furtherly, like support Sp164 and point mass Mass167 at point (11, 11, -82.5), which is consistent with the values in Figure2.5.

Name	Description	X [m]	Y [m]	Z [m]	X-Tra	Y-Tra	Z-Tra	X-Rot	Y-Rot	Z-Rot
Sp41	Support Point	-11	-11	-82.5	15402296.2 N/m	15402296.2 N/m	30892494 N/m	0 N*m	0 N*m	84752765.9 N*m
Sp82	Support Point	11	-11	-82.5	15402296.2 N/m	15402296.2 N/m	282912612 N/m	0 N*m	0 N*m	84752765.9 N*m
Sp123	Support Point	-11	11	-82.5	15402296.2 N/m	15402296.2 N/m	282912612 N/m	0 N*m	0 N*m	84752765.9 N*m
Sp164	Support Point	11	11	-82.5	15402296.2 N/m	15402296.2 N/m	282912612 N/m	0 N*m	0 N*m	84752765.9 N*m
Sp40	Support Point	-11	-11	-82.26	93698064 N/m	93698064 N/m	187930867 N/m	0 N*m	0 N*m	515583519 N*m

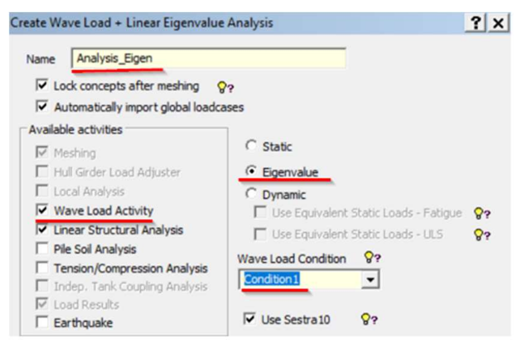
Name	Description	Mass [kg]	X [m]	Y [m]	Z [m]
Mass44	Point Mass Scalar	397.62	-11	-11	-82.5
Mass85	Point Mass Scalar	397.62	11	-11	-82.5
Mass126	Point Mass Scalar	397.62	-11	11	-82.5
Mass167	Point Mass Scalar	397.62	11	11	-82.5
Mass43	Point Mass Scalar	2418.87	-11	-11	-82.26

Figure 2.11 The imported linear springs and point masses

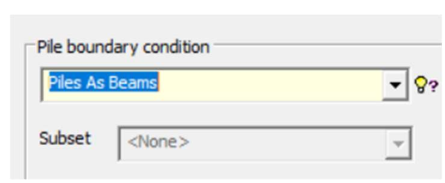
2.5 Execute Eigenvalue Analysis

With the piles and linearized springs, a linear analysis can be created and executed. Take eigenvalue analysis as an example. The steps include,

- Create a new analysis, select **Eigenvalue**, check **Wave Load Activity** and select **Condition1** as wave load condition.

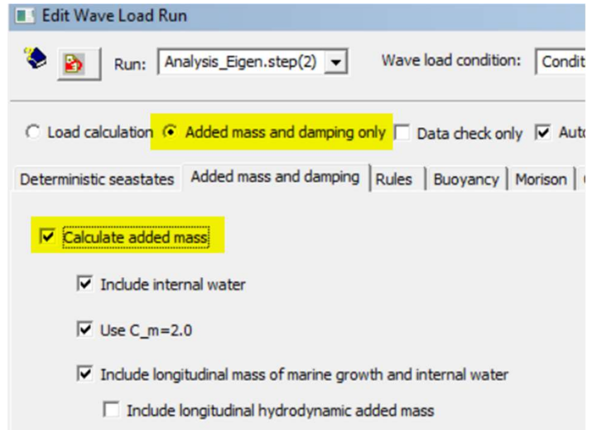


- Edit **Analysis_Eigen.step(1),** change Pile boundary condition to "**Piles As Beams**".



NOTE: the pile boundary condition "Piles As Beams" is very critical for a linear analysis with piles and linearized springs, this will make the pile element be meshed as beam element together with the linear springs.

- Edit **Analysis_Eigen.step(2),** check "**Added and damping only**" and then check "**Calculate added mass**". Click **OK** to accept these setting.



- Run the analysis.
- View the results in Sestra.lis file

Number;	Eigenvalue;	Frequency;	Period
1;	3.918428e+00;	3.150475e-01;	3.174125e+00
2;	3.938628e+00;	3.158585e-01;	3.165974e+00
3;	6.285558e+01;	1.261804e+00;	7.925158e-01
4;	7.745452e+01;	1.400695e+00;	7.139314e-01
5;	1.699455e+02;	2.074794e+00;	4.819756e-01
6;	3.157448e+02;	2.828058e+00;	3.535996e-01
7;	3.259379e+02;	2.873344e+00;	3.480266e-01
8;	4.523197e+02;	3.384877e+00;	2.954317e-01
9;	5.236037e+02;	3.641845e+00;	2.745861e-01
10;	6.666326e+02;	4.109258e+00;	2.433529e-01

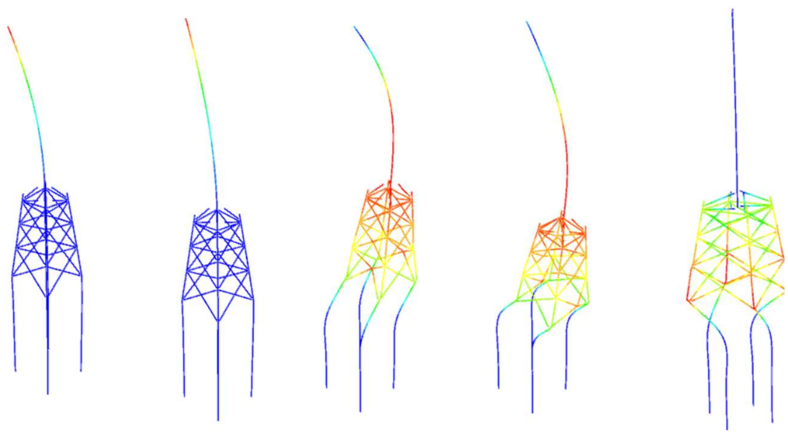


Figure 2.12 The eigen period and mode shapes of 1st – 5th

- Go to Mesh-All, the different FEM representatives in these two analyses are shown below.

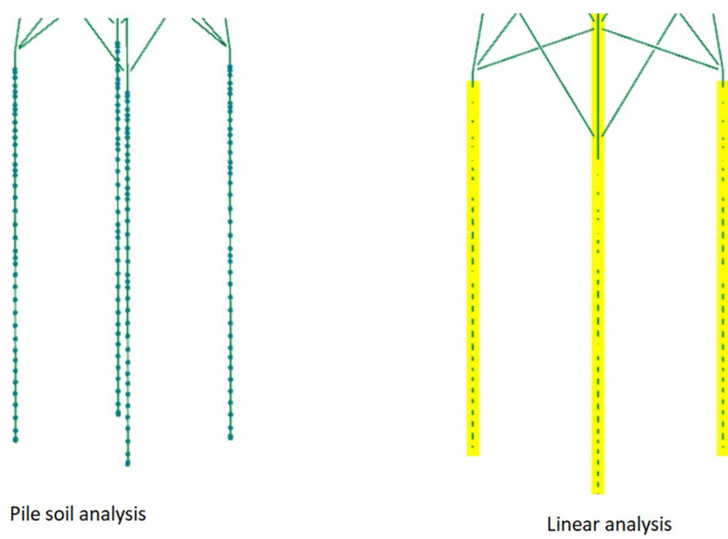
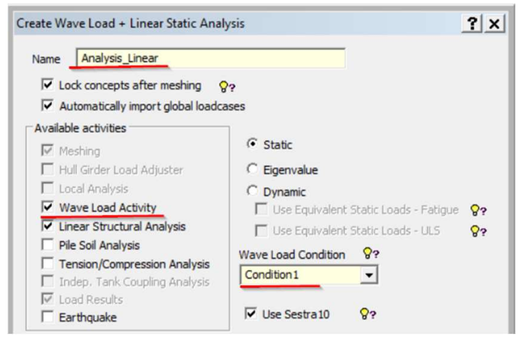


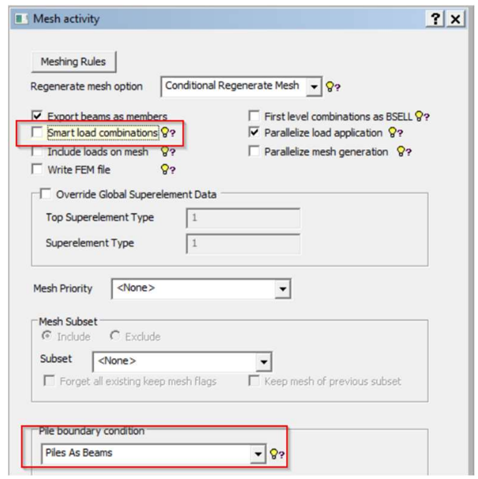
Figure 2.13 FEM model for pile soil analysis & linear analysis

2.6 Execute Linear Static Analysis

In this section, a linear static analysis will be created and the same load combination LComb_N090 will be used to compare the pilehead nodal displacement between pile soil analysis and linear analysis. The steps include,

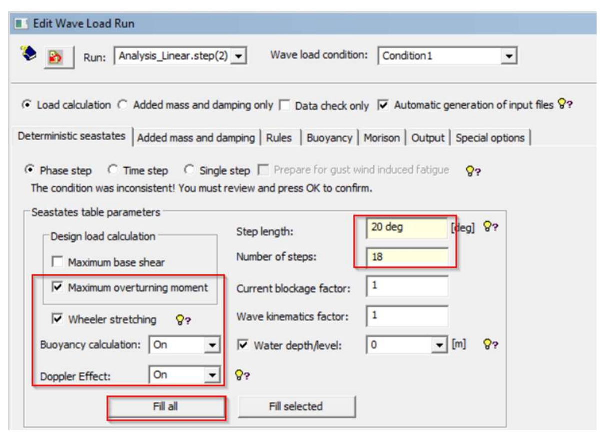


- Create a new analysis, check **Wave Load Activity** and select **Condtion1** as wave load condition.

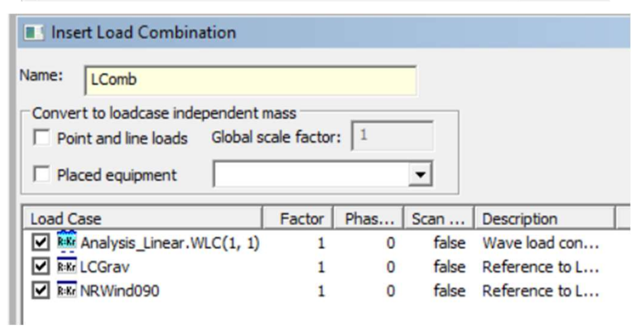


- Edit **Analysis_Linear.step(1),** uncheck "**Smart load combination**" and change Pile boundary condition to "**Piles As Beams**".

Note: Uncheck "smart load combination" in this step, then we can check load sum for load combination in sestra.lis, compare it with the load sum in pile soil analysis, to make sure that the same load is used for comparison of pilehead displacements.

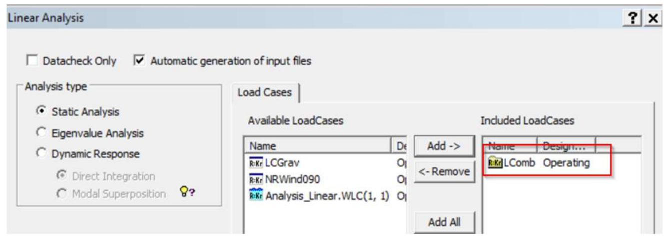


- Edit **Analysis_Linear.step(2),** check the options shown in the right picture and click **Fill all**. Click **OK** to accept all setting.



- Create a load combination **LComb** which is exactly same with LComb_N090 in pile soil analysis.

- Edit Edit **Analysis_Linear.step(3)**, to select **LComb** in the solution.



- Run the analysis
- Compare results between pile soil analysis and linear static analysis.
    - Compare the load sum to make sure that the same loads are used for comparison.

```
Load sum for all result cases (internal number):
   result case;            tx;            ty;            tz;            rx;            ry;            rz
             1;  5.846145e+02;  2.803250e+06; -1.461431e+07; -2.040887e+08;  2.037481e+06; -4.020529e+05
```

pile soil analysis

```
Load sum for all result cases (internal number):
   result case;            tx;            ty;            tz;            rx;            ry;            rz
             1;  5.846145e+02;  2.803250e+06; -2.174142e+07; -2.040887e+08;  2.037481e+06; -4.020529e+05
```

linear analysis

Figure 2.14 Load sum in pile soil analysis & linear analysis

Note: The difference of tz is because of the point masses below the mudline which came from the soil/fluid inside the pile defined in pile characteristics.

- Compare the pilehead node displacements, it shows that the displacement between pile soil analysis and linear analysis is comparable.

LComb_N090 : FEM Node Displacement

- Sorted by Name (Ascending)
- Then sorted by FEM Node (Ascending)
- Filtered by Limit : LimitInSet(PileheadJoints)
- Analysis : Analysis_Nonlinear

Name	FEM Node	X [m]	Y [m]	Z [m]	DX [m]	DY [m]	DZ [m]	RX [deg]	RY [deg]	RZ [deg]	Complex
Jt31	71	-11	-11	-27.5	0.00260282	0.0207982	0.00194138	-0.112124	0.0108726	0.00185002	No
Jt32	73	11	-11	-27.5	-0.0028235	0.0204314	0.00186857	-0.110528	-0.0120043	-0.00232798	No
Jt33	83	-11	11	-27.5	-0.00583361	0.0243623	-0.00838627	-0.123699	-0.0230028	0.00115811	No
Jt34	93	11	11	-27.5	0.00612384	0.0241424	-0.00847065	-0.122618	0.0245164	-0.00152128	No

LComb : FEM Node Displacement

- Sorted by Name (Ascending)
- Then sorted by FEM Node (Ascending)
- Filtered by Limit : LimitInSet(PileheadJoints)
- Analysis : Analysis_Linear

Name	FEM Node	X [m]	Y [m]	Z [m]	DX [m]	DY [m]	DZ [m]	RX [deg]	RY [deg]	RZ [deg]	Complex
Jt31	71	-11	-11	-27.5	0.00260437	0.0208258	0.0019415	-0.112147	0.010894	0.00186287	No
Jt32	73	11	-11	-27.5	-0.0028275	0.0204568	0.00186881	-0.11054	-0.0120391	-0.00235119	No
Jt33	83	-11	11	-27.5	-0.00581563	0.02431	-0.00839219	-0.12326	-0.0229633	0.00120421	No
Jt34	93	11	11	-27.5	0.00610422	0.0240885	-0.00847675	-0.122171	0.0244646	-0.00157406	No

Figure 2.15 Pillehead nodal displacement in pile soil analysis & linear analysis

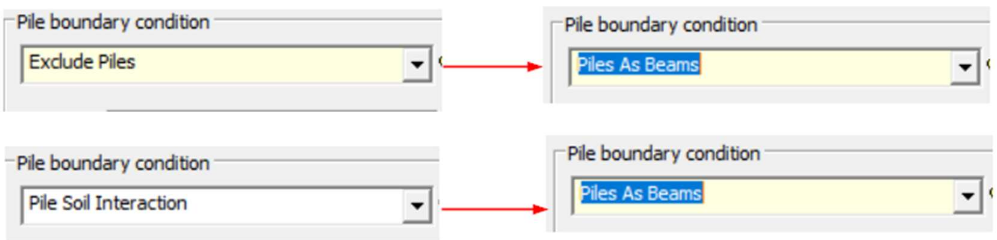
3 POSSIBLE ISSUES AND LIMITATIONS

3.1 Error in Factorization of the Stiffness Matrix

- In a linear analysis with piles and linearized springs along the pile, if the pile boundary condition under Edit Mesh Activity kept default “Pile Soil Interaction” or set as “Exclude Piles”, then the stiffness matrix error will be printed in the sestra.mlg file.

```
Execution failed!
    Error in factorization of the stiffness matrix. The matrix is singular to machine precision. Please verify
    sufficient boundary conditions and non-zero stiffness of all elements. If the model contains interconnected
    planar shell elements, please also verify that the stiffness singularity coefficient is not exactly zero. The
    problem may be related to the following node and/or elements. External node number 61. External element numbers
    64, 72, 120, 121, 128, 130. (Internal node number 61. Internal element numbers 64, 72, 120, 121, 128, 130.)
```

Solution: Change the pile boundary condition to “Piles As Beams”.

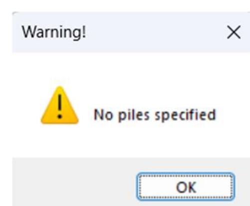


- After importing linearized springs along the pile, if add joints on the pile, the default connection setting is disconnecting pile from spring and point mass. Then the error in factorization of the stiffness matrix will be issued when executing linear analysis or result in huge displacement on the piles.

Solution: Delete all joints on the piles or modify the joint to connect pile with mass and spring.

3.2 No Piles Specified

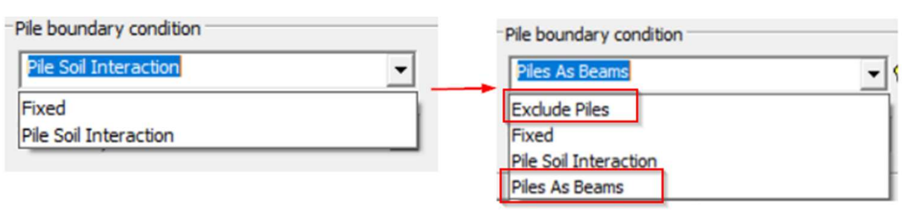
In the pile soil analysis, if the pile boundary condition is “Piles As Beams”, a warning window will pop up indicate that “No piles specified” when executing the analysis.



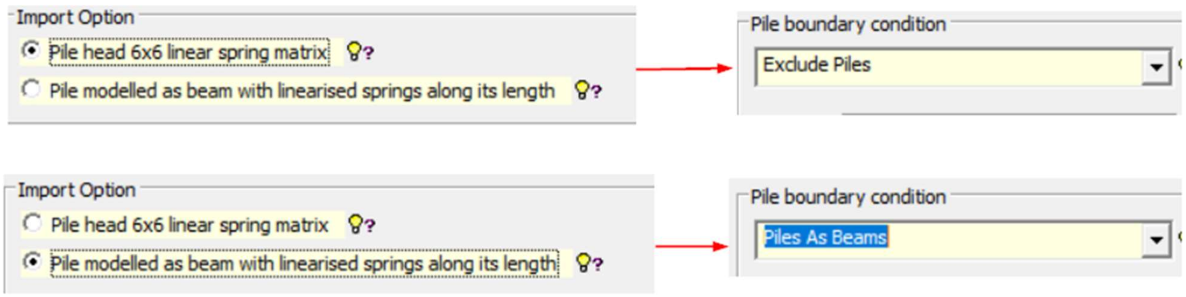
Solution: Edit mesh activity to change pile boundary condition from Piles As Beams to Pile Soil Interaction.

3.3 Pile Boundary Condition under Edit Mesh Activity

From Genie 8.8, another two pile boundary conditions under Edit Mesh Activity are introduced, that is “Exclude Piles” and “Pile As Beams”.



The two new pile boundary conditions are used in different import option. If the pilehead spring is imported, then the option “Exclude Piles” is needed. Otherwise, the option “Piles As Beams” is needed.



3.4 Switch off Pile Linearization Process

From GeniE8.8 and later versions, automatic pile-soil linearization along pile is activated by default. Then for each load case in the Sestra solution, the corresponding linearized springs along the pile nodes will be generated and stored in SPLICE_LINEAERISEDPILE_T1.FEM. Then for the analysis have a large number of load cases or a time domain analysis, there will be a problem, that is, program will spend a lot of time to accomplish this process.

Solution:

1) Switch off the pile linearization process by modifying Splice.inp to set MISC(1) = 0.

2) Edit pile soil analysis, go to Sestra > Load Cases to select the desired load cases into included load cases window to reduce the number of load cases in pile linearization process.

3.5 The effect from Self-weight Load in Load Combination

When compare the results between pile soil analysis and linear static analysis, the users should understand the difference between Splice and Sestra when handling the pile part.

In pile soil analysis - the load case of self-weight, or the load combination includes self-weight load case only always cover the weight of superstructure and will be solved by Sestra. The pile self-weight and the weight from internal soil/fluid is always be handled as a separate part and be solved by Splice which with load factor=1.0 in all load cases.

In linear static analysis with springs and piles as beams – the load case of self-weight, or the load combination includes self-weight load case always cover the whole structure.

So, when comparing pile displacements in two models, any load factor of self-weight load case rather than 1.0 (exclude self-weight load case or the load factor is not 1.0) in a load combination will lead to a bigger difference in axial displacement.

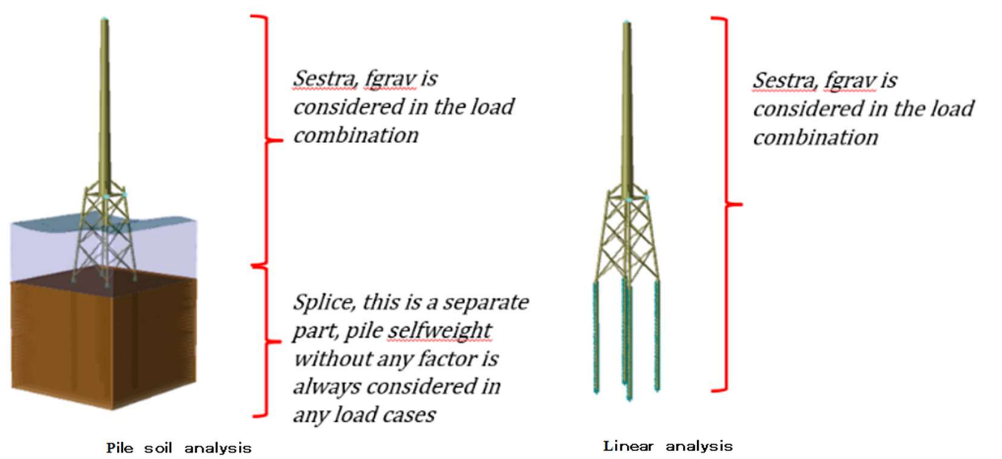


Figure 3.1 Different handle method for load factor of self-weight load

3.6 Different Eigenvalue Results between Two Linearized Springs

The methodology to extract pilehead equivalent stiffness matrix and linearized spring along the pile is different, which includes:

- The non-linear SPLICE stiffness matrices are softest in the direction of the applied lateral loading. The pilehead equivalent stiffness matrix does not contain this directional property. But the linearized spring along the pile contains this directional property.
- The pile head equivalent stiffness matrix is extracted by matching the pilehead displacement on one specified pile, so only the displacements on one pile need to be matched. But the linearized spring along the pile is derived from the solution of each pile, then the displacements on each pile need to be matched.

Then this will lead to the different stiffness of the structure. So, the eigenvalue results will be different more or less.

The bigger difference lies in the modal mass participation factor. If the number of the mode shapes are same, then the modal mass participation factor in the model with linearized springs along pile will be less than the model with pilehead equivalent stiffness matrix. This is because that the mass of the pile and fluid/soil inside the pile has been counted into the total mass, but the pile part will contribute to the higher order mode shapes.

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