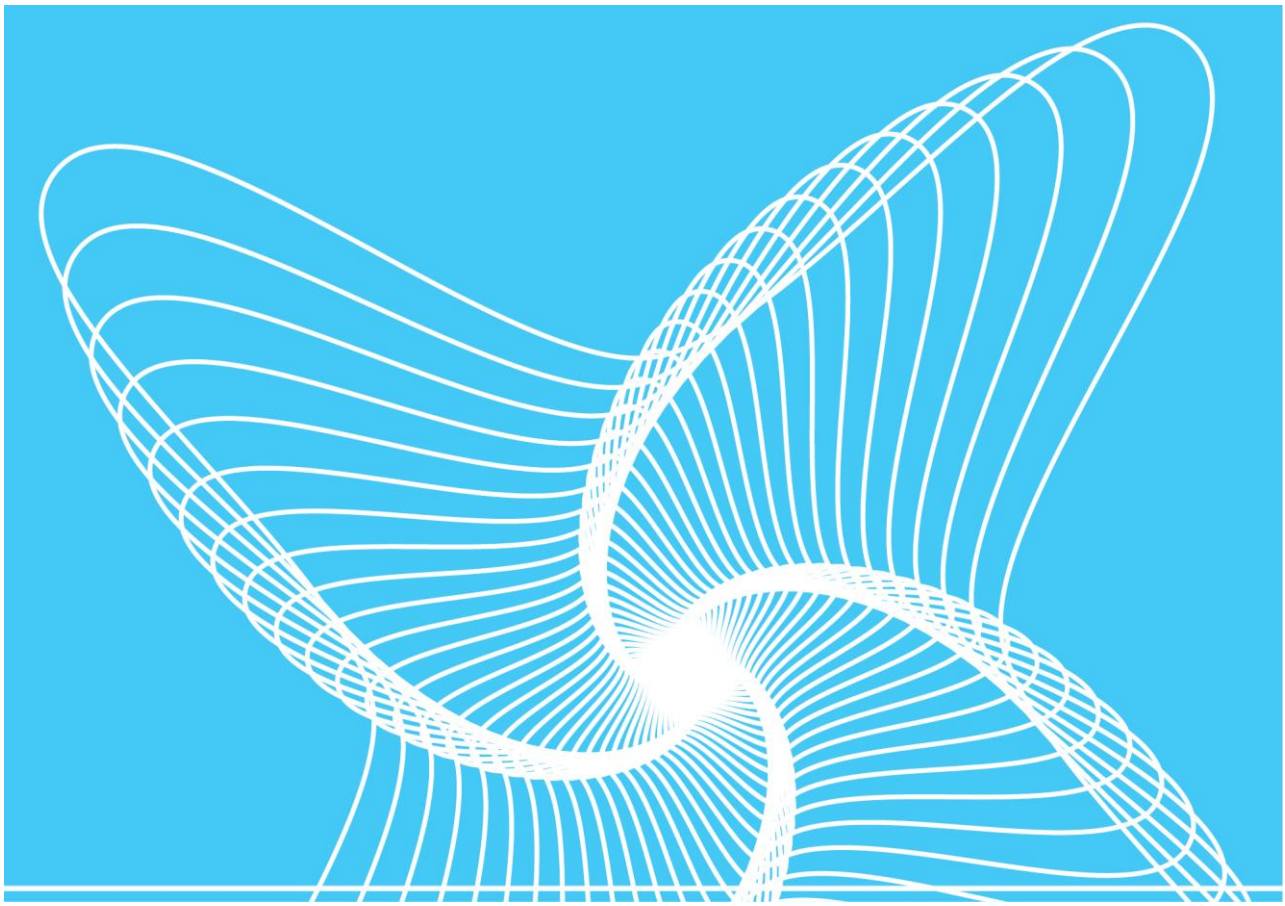
DNV·GL

SESAM USER MANUAL

GeniE

SACS import guidelines

Valid from program version V8.1



Sesam User Manual

GeniE - SACS import guidelines

Date: 15 April 2021

Valid from GeniE version V8.1

Prepared by DNV GL – Digital Solutions

GeniE to SACS import guidelines

Table of Contents

1. INTRODUCTION

The Sesam Fixed Structure Package has a powerful capability to import SACS models. The conversion is done from inside Sesam's GeniE program – in the following this program is referred to as GeniE. In addition, the Sesam programs Wajac, Splice and Sestra are referred to.

The functionality for importing SACS models includes data for wave loading and pile soil analysis. For details of SACS input data interpretation, see the "Sacs to Sesam" Reference Document. The reference document describes what can be imported as well as known limitations in data conversion from SACS.

Also read about import of SACS models in section 14.1.9 "Import SACS file" in the User Documentation.

Before you start any conversion: Note that the database and input units must be identical before starting to import a SACS model. Otherwise, data related to hydro properties and environment will be incorrect.

To import a SACS model should ideally be to "push the import button" followed by analysis and code checking – and often it is.

However, it may be necessary to adjust the default import options and/or tolerances to get the optimal import result. When importing soil data the possibility to refine soil layers should be utilized. Finally, the imported model may need to be modified and/or enhanced to make everything correct and suited for analysis.

This document describes challenges/problems that may occur related to SACS import and how to solve them. It is based on DNV GL Software experience from importing more than 100 models.

1.1 How to ensure correct import

A SACS import session may consist of only a SACS input file or both a SACS and a SACS PSI input file.

The SACS input file contains principal structural data, i.e. geometry, material and physical properties as well as nodal and element loads. Additionally, it contains key hydrodynamic loading data including wave and current specifications and flooded members.

The SACS PSI input file contains definition of piles and soil. The soil part defines both soil layers and soil properties (soil curves).

Note that even if the session consists of both file types it is advisable to start with importing the SACS input file only, ref. options in section 1.1.3. A static linear analysis, possibly including a wave load analysis, should be performed. It may be necessary to do modifications to make the analysis execute successfully based on messages given during import, Sestra error messages and/or Wajac error messages. The guidelines given in this document should help to decide what modifications are necessary.

When the static linear analysis executes successfully the model including pile and soil data should be imported. A pile soil analysis, possibly including a wave load analysis, should be performed. Additional modifications of the model based on error messages from Gensod, Splice and/or Sestra may be needed. Also, for these cases this document should help to decide what modifications are necessary.

Note the following regarding model size limitations.

- Max number of nodes and elements, respectively, is 75000.
- Max number of load cases: This is flexible, but the initial number is 4000
- Max number of basic loads per load case: This is flexible, but the initial number is 75000
- Max number of load combinations: 3000
- Max number of load cases per load combination: 100

The product of number of load cases times number of basic loads per load case must be less than 4000 * 75000.

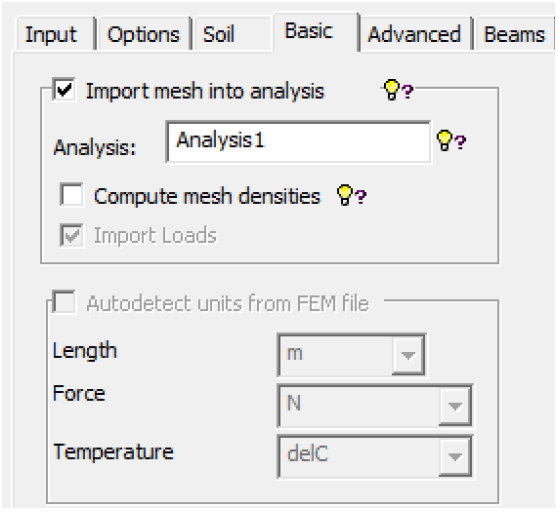
1.1.1 Structural data import

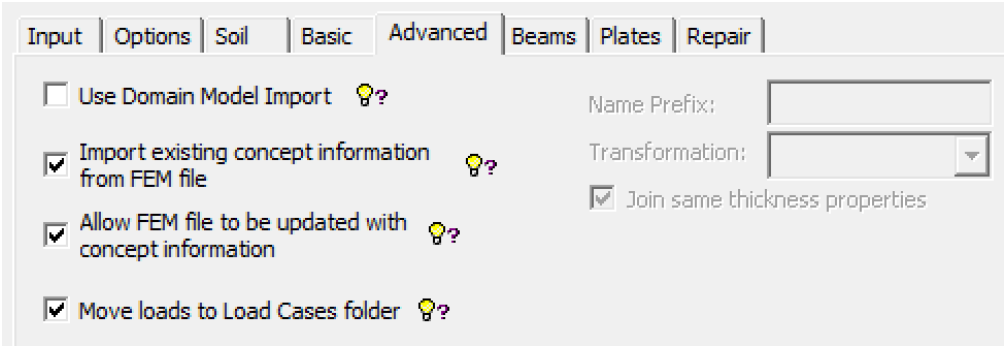
By default both “Import mesh into analysis” and “Import existing concept information from FEM file” options are checked, as shown in Figure 1. If the “Import existing concept information from FEM file” option is unchecked beam concepts are not imported. By default, joint concepts are not imported. To import joint concepts the “Import named joints” option must be checked.

By checking the ‘Use SACS beam name’ option SACS naming for beams is preserved, i.e. start and end joint name is appended. This is useful when comparing SACS and GeniE models, e.g. when verifying code check results.

By default, each GRUP card of the SACS input file is imported as a named set. Typically, these sets represent cross sections. If unchecking “Import GRUP cards as named sets” only the utility sets are imported, see chapters 5.4 to 5.8. Unchecking the option may give significant performance gain.

If the “Import mesh into analysis” option is unchecked a Name Prefix may be given. However, if the SACS input file contains load combinations they will not be correctly handled. It is therefore advised to avoid using Name Prefix when the “Import mesh into analysis” option is unchecked.





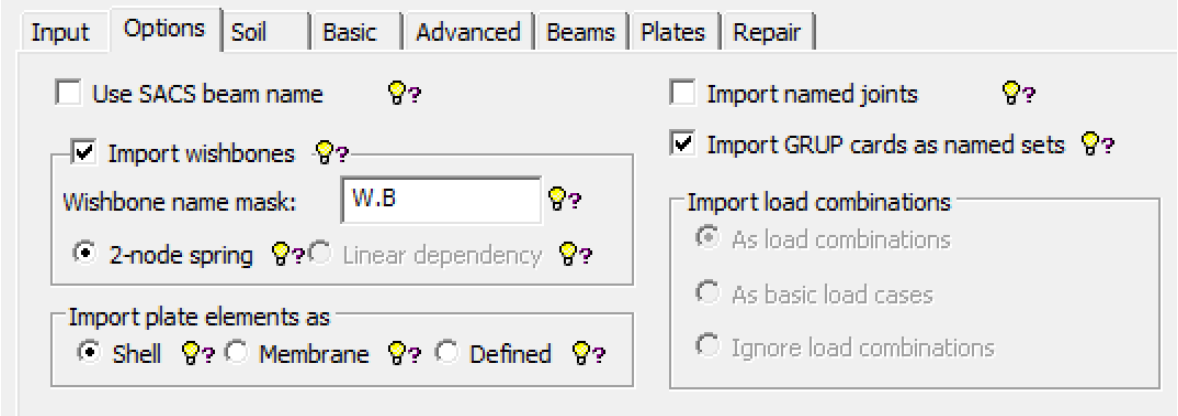


Figure 1: The import Options tab

The "Model Repair" option is by default checked, as shown in **Figure 2**. For a description of this feature see the GeniE User's Guide, Volume 3, chapter 10.4.4. As described there it will force beams to be co-linear by moving end-coordinates within the given angular and length tolerances. In particular, this functionality may be important for jacket models with complicated joint areas and inner/outer beam (pile/leg) structure. Note that the modified end-coordinates may result in inaccurate load values for imported point loads on beams. For models without overlapping beams it may be advisable to uncheck the "Enable Model Repair" option in the Repair tab of the "Import SACS file".

For GeniE version 6.6 and later versions the default angular tolerance is 2 deg. However, for older versions the default value is 0 deg. Experiences have shown that this may result in an inconsistent structure. For versions older than 6.6 it is advised to set angular tolerance to 2 degrees before importing, e.g. by the scripting command:

```
GenieRules.Tolerances.angleTolerance = 2 deg;
```

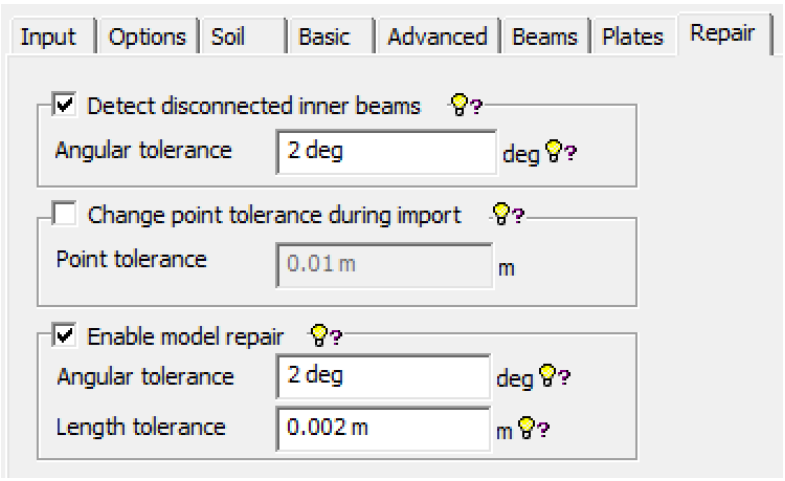


Figure 2: The import Tolerances tab

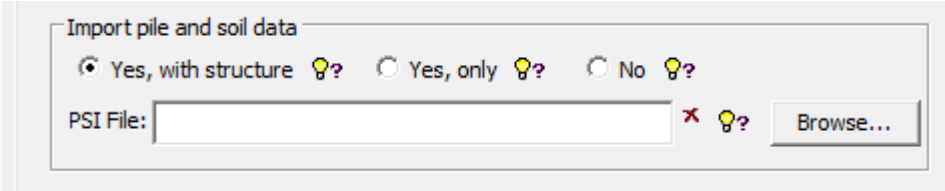
1.1.2 Wave load data import

Both "Import wave load data" (Input tab) and "Import mesh into analysis" (Basic tab) options are by default checked. It is possible to uncheck the "Import mesh into analysis" option when importing wave load data, but it is not recommended. In this case load combinations that contain wave load cases these will not be handled.

1.1.3 Pile and Soil data import

The “Refine soil layers” (Soil tab) is unchecked by default. It is recommended to check this option to control the layer thickness. In general an element length of 1 meter is recommended. The user can also define the layer thickness based on SACS PSI input file.

Note that there are three options with respect to import the PSI data:



Yes, with structure: The piles and soil data will be converted and imported immediately after the model geometry and loads have been established.

Yes, only: Use this option when the piles and soil data are converted and imported in a separate session after the model geometry and loads are established. This is typically done when the user wants to verify the model by performing a static analysis on fixed supports prior to running the pile-soil iteration analysis. Note that the input file used when importing the geometry and loads must also be selected when using this option.

No: Do not import piles and soil data.

1.2 Where to find information

When importing a SACS input file, a LOG file is generated. The content of this file is routed to the *Message tab* of GeniE. If problems occur during import a message of type

```
SacsImporter.DoImport("C:/DNV/Workspaces/GeniE/jacket/sacinp.inp");
// Please check Messages area for 6 warnings, 1 information message.
```

appears in the *Command Line tab*. The *Message tab* will contain these 6 warnings and 1 information message.

Note that the text buffer of the *Message tab* is limited and parts of the information may be lost. However, the LOG file contains all information and should be inspected. Additional information is found in the CHK file, e.g. more details about unsupported data, wishbone element information, relation tables between SACS and GeniE objects (joint, member and plates) and a cross section table.

The name of the LOG and CHK files are <filename>.LOG and <filename>.CHK, where <filename> equals the SACS input file name. In this example the file names are ‘sacinp.LOG’ and ‘sacinp.CHK’.

If wave load data is imported a command input file named ‘WajacToGeniE.js’ is generated and read. In addition to command syntax, this file may contain warning and error messages that are routed to the *Message tab*. If problems occur during wave load data import a message of type

```
WajacImporter.DoImport("C:/DNV/Workspaces/GeniE/jacket/sacstmp_Wajac.INP");
// Please check Messages area for 16 warnings.
```

appears in the *Command Line tab*. The *Message tab* will contain these 16 warnings. For the same reason as mentioned above with regard to the text buffer, the ‘WajacToGeniE.js’ file could be inspected when having problems related to import of wave load data.

Correspondingly, if importing pile and soil data from the SACS PSI input file, a command input file named ‘SacsPSIToGeniE.js’ is generated and read. In addition to command syntax, this file may contain warning and error messages that are routed to the *Message tab*. If problems occur during pile and soil data import a message of type

```
SacsPSIImporter.DoImport("C:/DNV/Workspaces/GeniE/jacket/psiinp.inp");
// Please check Messages area for 1 warning.
```

appears in the *Command Line tab*. The *Message tab* will contain this 1 warning. For the same reason as mentioned above with regard to the text buffer, the 'SacsPSIToGeniE.js' could be inspected when having problems related to import of pile and soil data.

The same as described for import of pile and soil data is valid for import of weight data and wind area data. 'SacsWeightsToGenie.js' and 'SacsWindToGenie.js' command input files are generated and may contain warning and error messages that are routed to the *Message tab*. If problems occur during import similar information appears in the *Command Line tab* related to `SacsWeightsImporter` or `SacsWindImporter`.

2. DURING IMPORT

2.1 Importing the SACS input file

This chapter describes problems that may occur when importing a SACS input file.

2.1.1 Explicitly defined cross section not supported

Messages as shown below appears in the Message tab/LOG file.

```
Warning: Section not found or not supported, beam  555 ,section W24X76A
Warning: Section not found or not supported, beam  556 ,section W24X76A
```

What to do:

Inspect the SACS input file and look for the cross section named W24X76A.

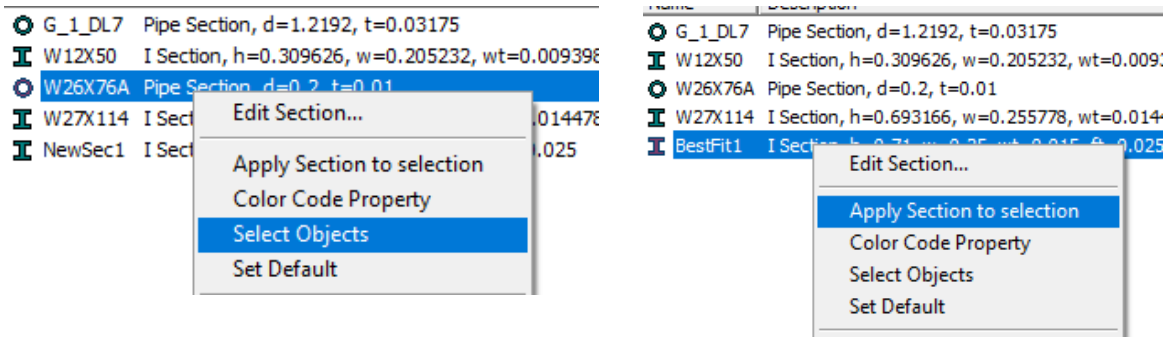
```
SECT W24X76A    PGB                                  22.83 1.72760.71 1.1181.2701.20
```

Look up “Reference Documents” from “Help→Help Topics…” and select “Sacs to Sesam”. Find the description of the SECT card and observe that cross section type PGB (Boxed Plate Girder) is not supported.

In this case the user can redefine the cross section (e.g. to a TUB) the cross section not supported and read the SACS input once more.

```
* CT W24X76A    PGB                                  22.83 1.72760.71 1.1181.2701.20
SECT W26X76A    TUB                                  20.00 1.0
```

In GeniE a “best fit” to the not supported cross section must be define and assigned the relevant beams.



Note that the missing section property on beams also will be reported in the Structure Verification tool as described in chapter 3.1

2.1.2 Cross section in section library not supported

Messages as shown below appears in the Message tab/LOG file.

```
Warning: Section not found or not supported, beam 1421 ,section C6X10
Warning: Section not found or not supported, beam 1422 ,section C6X10
```

What to do:

Inspect the SACS input file and look for the cross section named C6X10. In this case no such SECT card is found. The cross section is not explicitly defined and should be part of the section library.

Look up “Reference Documents” from “Help→Help Topics…” and select “Sacs to Sesam”. Find the part that describes profiles supported by the built-in library. Observe that cross section C6X105 is supported.

In this case a reasonable fix is to change C6X10 to C6X105 in the SACS input file and do the SACS import once more. Alternatively, do as described in 2.1.1.

Note that the missing section property on beams also will be reported in the Structure Verification tool as described in 3.1

2.1.3 Wishbone elements not recognized

The status of wishbone import is reported in the Message tab and could look like as shown below.

```
       Reading members
Warning:    54 members rejected
Info:    18 wishbone elements detected
      Finished reading 2602 members
```

Note the warning about 54 members rejected. This might be an indication of wishbone elements not imported. In this case the CHK file should be inspected.

Wishbone elements are primarily identified by a name mask with default value "W.B". In addition a typical fixation and the length of the element are used as indicators. If one or both of fixation and length indicators are fulfilled, but the element name does not match the name mask, messages as shown below appears in the CHK file.

```
Warning: member with group label W.C is a wishbone candidate but does not comply with
wishbone name mask W.B. Member is rejected.
MEMBER1151F151C W.CSK 000000100111           F
Warning: member with group label W.D is a wishbone candidate but does not comply with
wishbone name mask W.B. Member is rejected.
MEMBER1387F387C W.DSK 000000100111           F
```

Note that this information does not appear in the Message tab.

What to do:

Redo the import with updated 'Wishbone name mask'. In this example the updated name mask should be '`W.B;W.C;W.D`'. Now the status reported in the Message tab is

```
       Reading members
Info:    72 wishbone elements detected
      Finished reading 2656 members
```

And the messages in the CHK file is only of type

```
Info: member with length    0.00000 and group label W.C is identified as a wishbone
element.
MEMBER1151F151C W.CSK 000000100111           F
```

2.1.4 Wishbone elements not accepted

Even if the SACS input file contains wishbone elements the Message tab may report that no wishbone elements are detected. In this case the wishbone name mask was modified to 'W.C' based on inspection of the SACS input file before importing.

```
       Reading members
Warning:    16 members rejected
      Finished reading 2820 members
```

Note the warning about 16 members rejected. Once again, this might be an indication of wishbone elements not imported.

When inspecting the CHK file 16 of the following messages are shown.

```
Warning: member with group label W.C is a wishbone candidate but does not comply with
fixity requirements. Member is rejected.
```

```
MEMBER1 641 611 W.CSK 000000100011
```

This states that the wishbone elements were rejected because they failed to fulfill the fixity requirement.

What to do:

Modify the “illegal” wishbone elements in the SACS input file. In the example above change the fixation as shown.

```
MEMBER1 641 611 W.CSK 000000100111
```

After the model is imported identify the corresponding springs, e.g. by selecting the set ‘W_C’ that corresponds to the ‘W.C’ GRUP. Reset spring stiffness to the original fixation for selected springs.

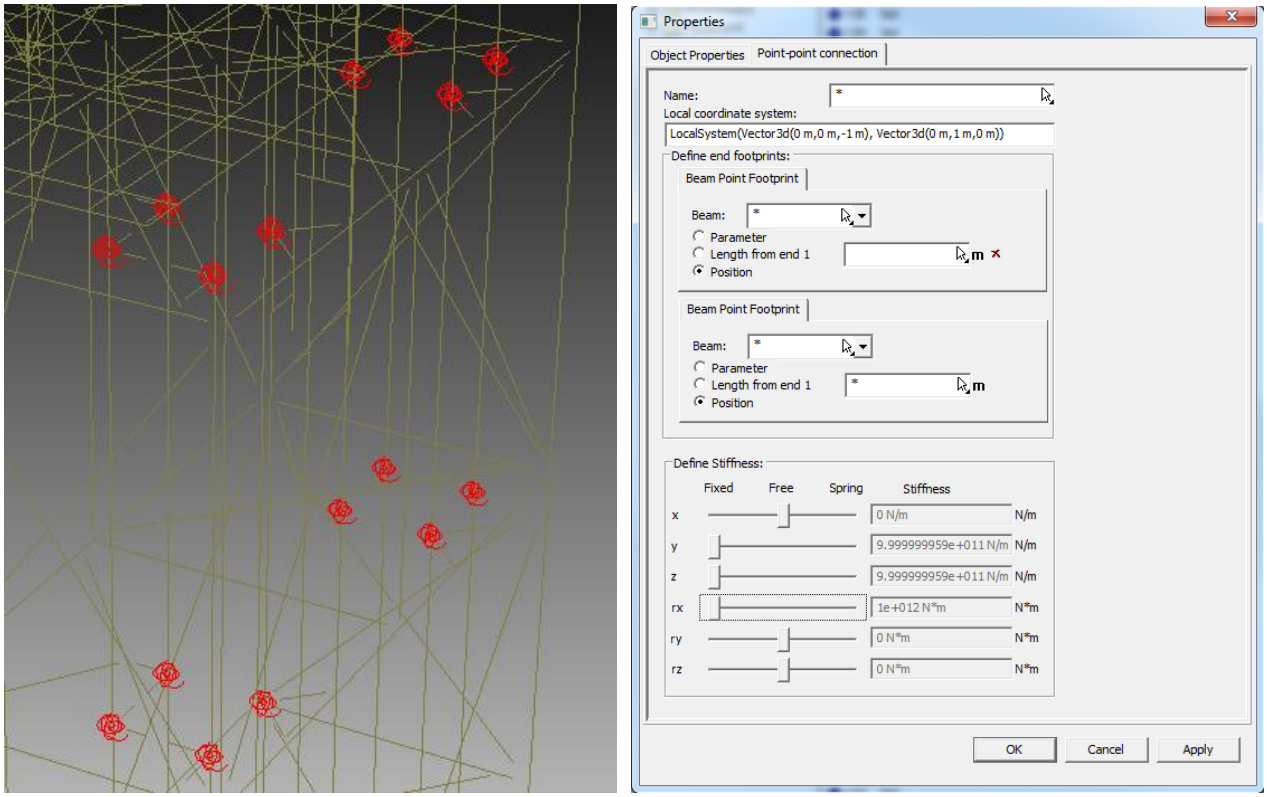


Figure 3: Select corresponding springs and reset to original fixation

2.1.5 Element not imported

Messages as shown below may appear in the Message tab/LOG file.

```
Warning:
```

```
Number of problem elements: 1
```

```
Elements not imported:
```

```
  3669
```

This message states that an element has been rejected; most likely due to that it is shorter than the import tolerance. This tolerance is by default 10 mm.

What to to:

From the sacstmpT1.FEM file the element length may be found. The error may be omitted by reducing the point tolerance during import to be smaller than element length, see Figure 4 below.

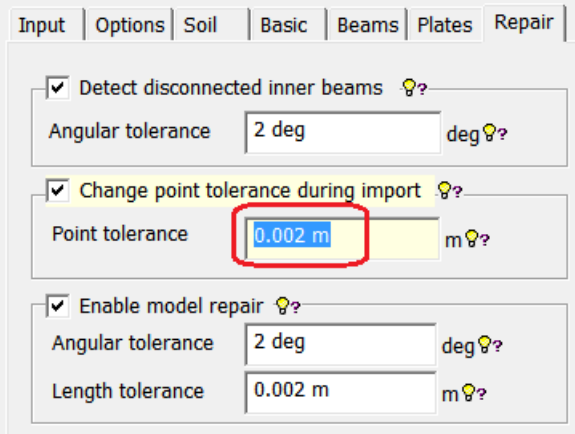


Figure 4: Reduce the point tolerance to be smaller than element length

To find the element length from the sacstmpT1.FEM file the following steps must be taken.

1. Locate the correct GELMNT1 card

```
GELMNT1    3.66900000E+03  3.66900000E+03  1.50000000E+01  0.00000000E+00
           5.62000000E+02  4.27400000E+03  0.00000000E+00  0.00000000E+00
```

2. Locate the relevant GCOORD cards

```
GCOORD     5.62000000E+02  1.46300011E+01  7.62000036E+00  1.37160006E+01
GCOORD     4.27400000E+03  1.46330004E+01  7.62000036E+00  1.37160006E+01
```

3. Calculate length by giving the following commands in the Command Line tab of GeniE

```
p1 = Point(1.46300011E+01,7.62000036E+00,1.37160006E+01);
p2 = Point(1.46330004E+01,7.62000036E+00,1.37160006E+01);
v1 = Vector3d(p1,p2);
print(v1.length());
-> 0.0029993 m
```

2.1.6 Plate offsets not handled

When plate offsets are imported messages as shown below appears.

```
Warning: Plate offsets cannot be converted !
PLATE  OFFSETS                      -8.9        -10.0 -8.9        -10.0
Warning: Plate offsets cannot be converted !
PLATE  OFFSETS                      -8.9
Warning: Plate offsets cannot be converted !
PLATE  OFFSETS                      -8.9        -10.0 -8.9        -10.0
Warning: Plate offsets cannot be converted !
PLATE  OFFSETS                      -8.9
      Finished reading    4 plates
```

Plates with offset are not supported in GeniE. However, the plates are imported without offset.

Misplaced plates may have several unwanted effects. One is that unexpected structural intersections can result in short edges and/or sliver faces. This again can give a bad, or even illegal, mesh. Also, loads may be inaccurately applied, resulting in inaccurate or even wrong results.

What to do:

There is no simple way of solving this problem. If plates are imported with some of the signs described one or more of the following actions may need to be taken.

1. For GeniE 6.9 and older versions, redo the import with the “Element:Plate 1:1” option checked. For GeniE 7.0 and later versions, uncheck “Allow joining plate elements”. In this case one plate is created in GeniE for each Sacs PLATE.
2. Delete plates that are causing problems with or without re-import.
3. Re-model plate area, possibly using rigid beams to represent eccentricities.

2.1.7 ACIS error 15067 – edge split at one end

During import dialogs indicating an ACIS error related to the geometry may appear. Such messages may often be ignored. However, a ***Tools|Structure|Verify*** from GeniE should be executed.

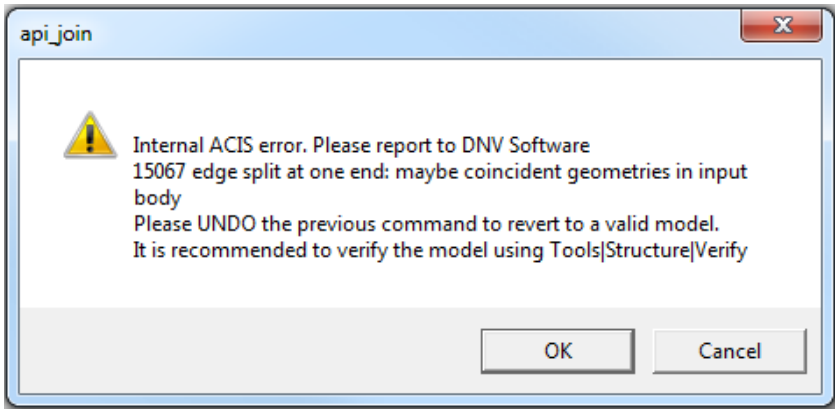


Figure 5: Dialog box indicating an error related to geometry

What to do:

The Structure Verify tool may point to a particular problem.

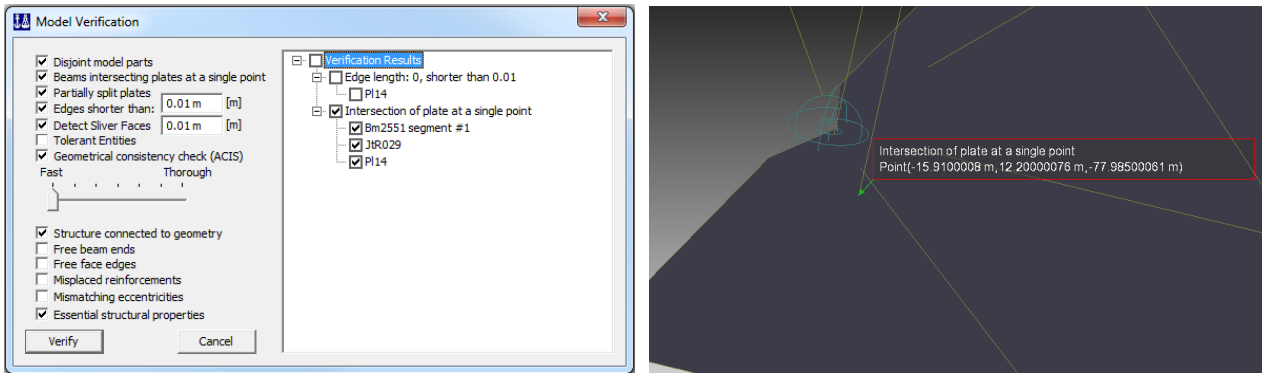


Figure 6: The Structure Verification tool may point to the geometrical problem

In this case the Bm2551 must be forced to snap to the adjacent beam by using the Move End functionality.

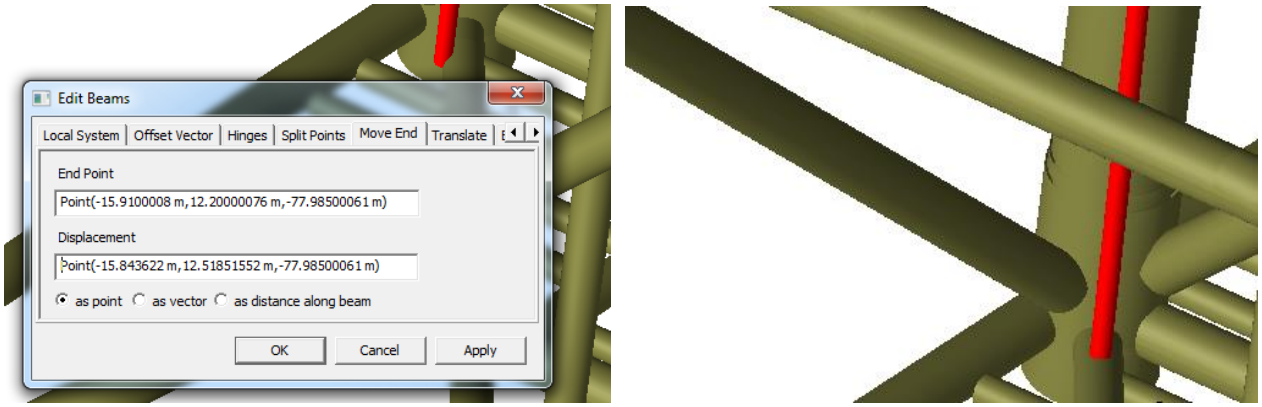
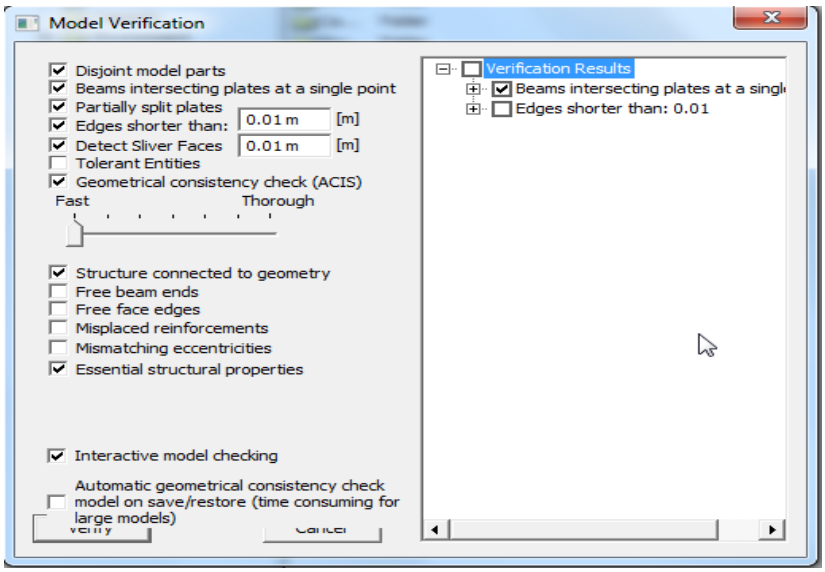
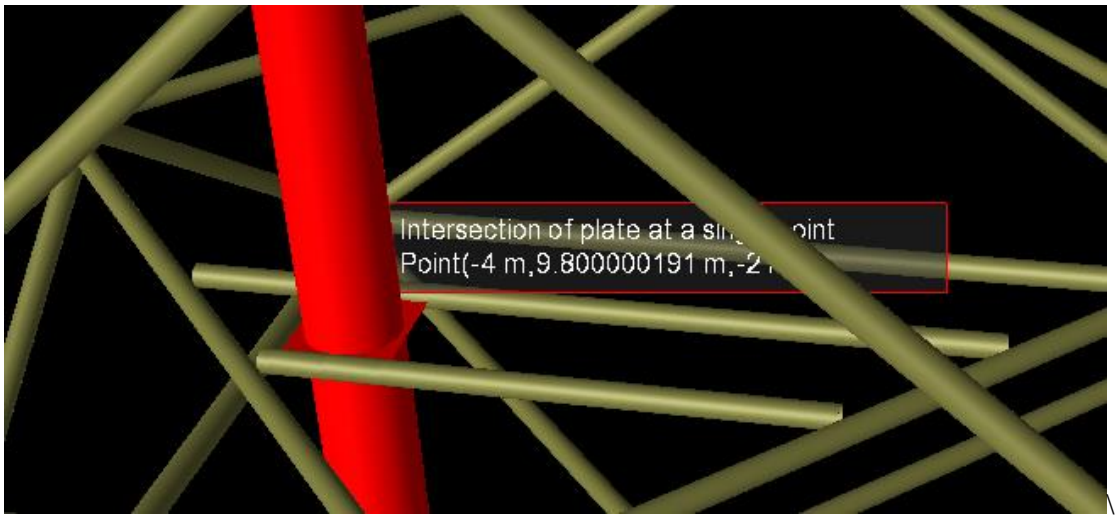


Figure 7: Force problem beam to snap to adjacent beam

2.1.8 Beam intersecting plate at a single point

As shown below, after importing a beam is surrounded by a small plate. When verifying structure it will show “Beam intersecting plate at a single point”. This will cause mesh problems.





What to do:

Delete the plate and connect the beam to the frame structure by some dummy beams or master-slave nodes.

2.1.9 ACIS error 15041 – loop does not contain given vertex

When the error message shown below appears during import an edge will not be created and should not be ignored. Note that this error has been reported to Spatial and registered as a bug. The fix will be integrated into GeniE ASAP.

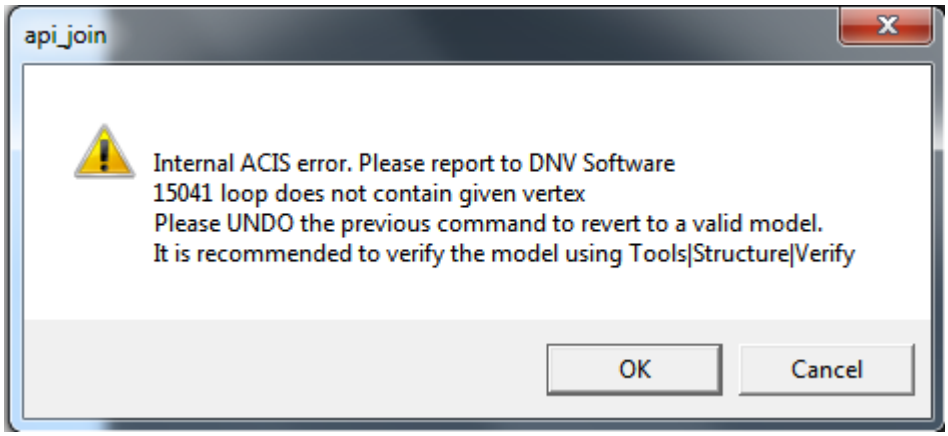
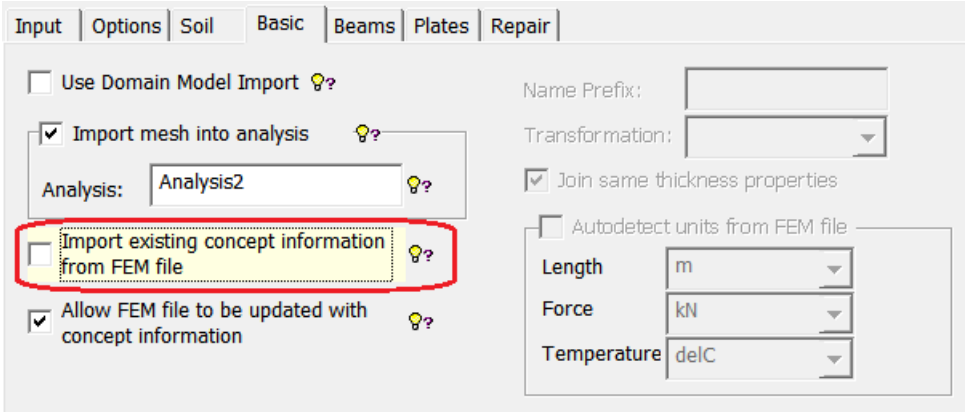


Figure 8: Dialog box indicating that an edge has not been created

Workaround

A possible workaround is to import the model without concept information, see figure below.



2.1.10 Wind area data or weight area data import fails

Both wind area data and weight data are imported via a js-file that is generated from the SACS input file. The wind area data js-file is named “SacsWindToGeniE.js” and the weight data js-file is named “SacsWeightsToGeniE.js”. If the import of one of these fails

- That particular data is not imported
- The rest of the import process is influenced

Below is shown an example where import of wind area data fails.

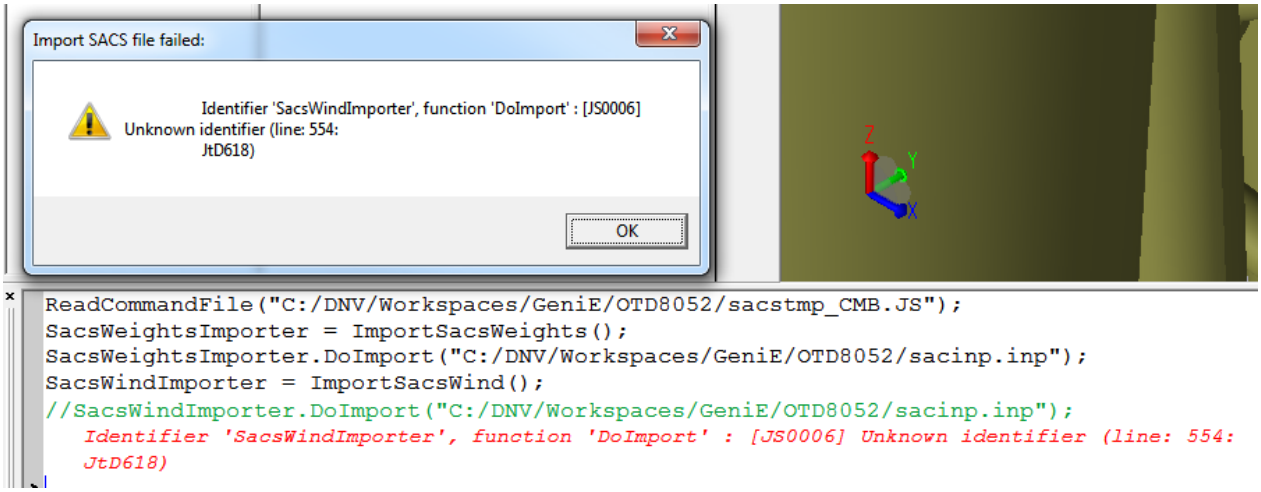


Figure 9: Wind area import fails due to joint not properly connected to structure

By analysing the "SacsWindToGeniE.js" file (line 554) and looking up joint D618 in the SACS input file we find that the joint is not properly connected to the structure. More specifically it is positioned in the interior of a plate. This is not handled by GeniE and no joint is created. The following procedure may be used to solve this and similar kinds of problems, both with regard to wind area data and weight data import.

1. Repeat the import process with the "Import wind area data" option unchecked. This time all data, except wind area data, is correctly imported.
2. Manually edit "SacsWindToGeniE.js". The undefined joint is part of the named set "Wind_EO_Set". Comment out adding joint D618.

   ```
   // Wind_ EO _Set.add(D618);
   ```

   Note that if D618 was the only joint of the set, everything related to "Wind_EO" should be deleted.
3. Read the corrected "SacsWindToGeniE.js" file manually as a post-import step.
4. Possibly repeat steps 2 and 3 if new errors occur during read.

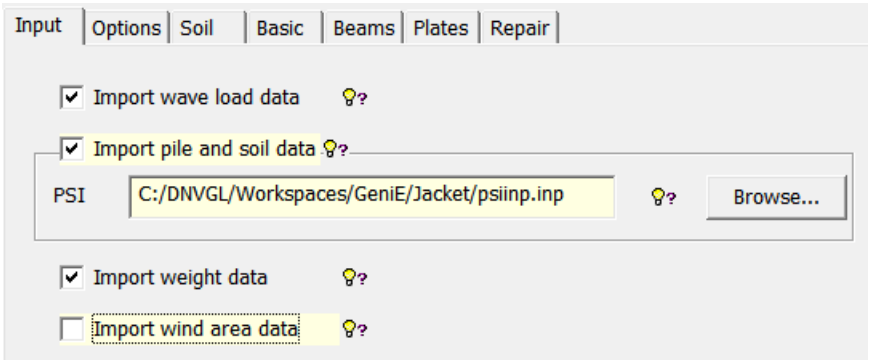


Figure 10: Repeat import process with "Import wind area data" unchecked

An alternative solution is to add the joint with legal coordinates at the top of the "SacsWindToGeniE.js" file.

A third alternative would be to edit the SACS input file to ensure that the coordinates of D618 is defined at a point in the structure where GeniE will accept a joint.

2.1.11 Load combinations could not be imported

If "Import wave load data" is unchecked the import may fail with the message "Load combinations could not be imported". This will happen if the SACS input file contains wave load data. The converter will create a load combination file (sacstmp_CMB.JS) that contains wave load cases independent of the status of the "Import wave load data" option.

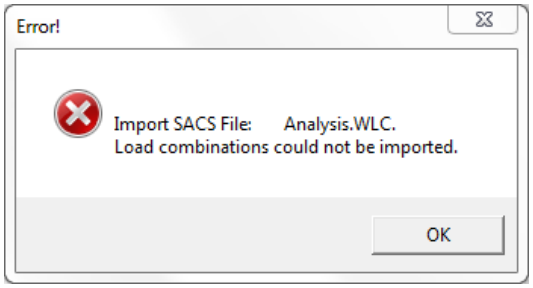


Figure 11: Load combinations could not be import

If this error message occurs the following steps are required:

- Repeat the import process with the “Ignore load combinations” option (Options tab) checked
- After completion of the import open the sacstmp_CMB.JS file in an editor and remove all references to wave load cases
- Read the sacstmp_CMB.JS file manually into GeniE as post-import step

Note that the file sacstmp_CMB.JS is generated independent of what “Import load combinations” option that is selected.

2.1.12 Unconnected joints are deleted during import

The following problem may occur in GeniE versions V6.5-04 to V6.9-05 when the “Import named joints” option is checked. During SACS import unconnected joints may be detected and deleted. Because the delete operation is not properly done, the import process may fail with an error message similar to:

```
//SacsImporter.DoImport("C:/DNV/Workspaces/GeniE/DF1/DF1-1 WHPF-JKT-DD-IP-001B-sac_revised2.inp");
   Identifier 'SacsImporter', function 'DoImport' : Unknown error
// Please check Messages area for 19 warnings, 1 information message.
```

The message tab will typically contain the following information:

```
Number of problem joints: 32
Nodes not imported as joints:  1262 1263 1264 1265 1266 1267 1268 1269
1270 1271 1272 1273 1274 1275 1276 1277 1278 1279 1280 1281
1282 1283 1284 1285 1286 1287 1288 1289 1290 1291 1292 1293
```

Workaround
Uncheck the “Import named joints” option.

2.1.13 Import is terminated due to problem reading FEM file

The following problem may occur in GeniE version V7.0 and newer due to problems reading the FEM file generated during SACS import, “sacstmpT1.FEM”. The following error message is shown.

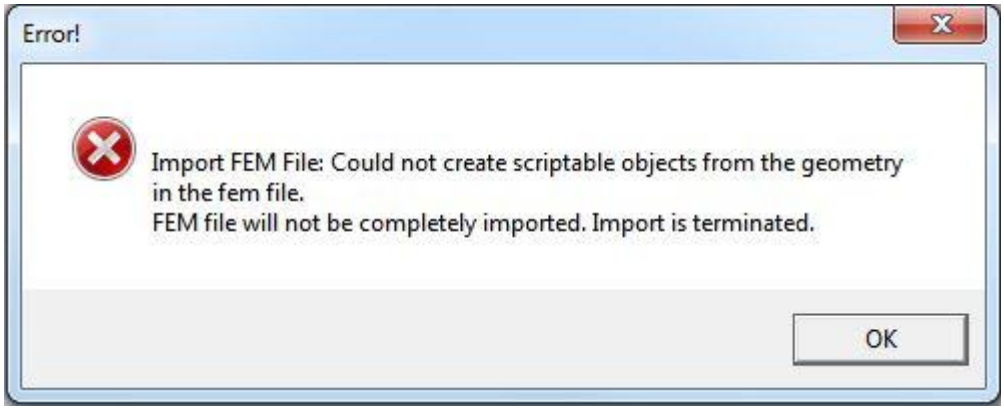


Figure 12: Import process fails due to problems reading the sacstmpT1.FEM file

Typically this problem is related to geometrical problems of the model and messages as shown below may be observed.

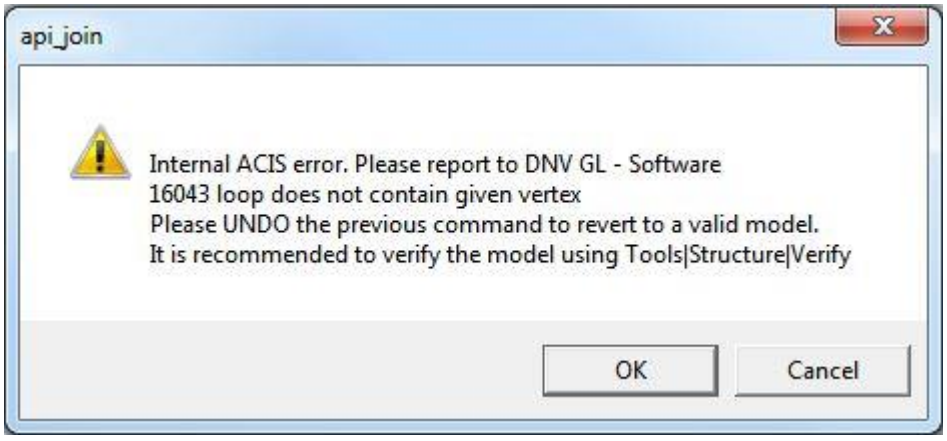


Figure 13: The problem is typically related to geometrical inaccuracies

Due to that the import is terminated, the model will miss essential information like hydro dynamic properties, environment data (water, current and soil), weights, wave and wind loads.

Workaround

Typically this problem originates from the “Model repair” part of the FEM import. This process is important for models having overlapping beams, i.e. legs with inner piles. If the model does not have overlapping beams the “Enable model repair” option should be unchecked before starting the import.

Even if the model has overlapping beams and this kind of problems occur the values, angular and length tolerance, should be reduced in an attempt to prevent the import from failing.

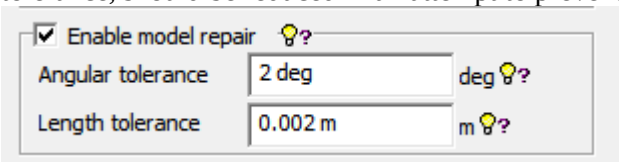


Figure 14: To prevent import from failing either uncheck "Enable Model repair" or reduce tolerances

Additionally, if this is a model with legs and inner piles, it is also advisable to switch off the detection of inner beams. The inner beams must then manually be assigned the wanted connection to the leg member after import.

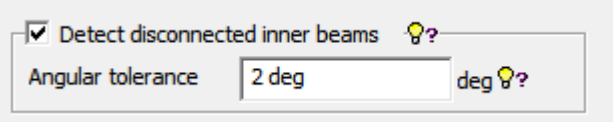


Figure 15: It is advisable to switch off detection of inner beams

Check that the inner beam is actually connected to relevant structural joints (select joint, RMB and select Properties, check the Connection Point definition. The inner beam should be connected to a connection point separate from the leg and brace members). If not, the inner beam needs to be deleted and re-modelled.

2.1.14 Wishbone elements – merging joints

Wishbone elements may have joints with different coordinates at start and end joint. When this distance is less than the "Point tolerance during import" the converter will "merge" the joints, i.e. the end joint (joint 2) will get the same coordinates as the start joint (joint 1).

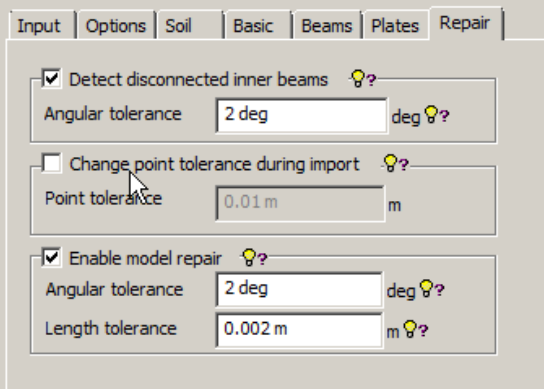


Figure 16 Point tolerance during import

Note that when the wishbone member has member offset, these offset values are used to define the local coordinate system (direction of local x-axis) for the two-node spring created. If no member offset is given the coordinates for start and end joints will be used.

2.1.15 Cross section DTB (Dented TuBular member)

Dented tubular member cross sections are converted and imported as tubular sections without any dent or grout information. Check for Warning message on the LOG file, typically:

```
Warning: SECT DTB1    is a dented tubular, all dent info will be skipped
```

2.2 Importing the SACS PSI input file

This chapter describes problems that may occur when importing pile and soil data from a SACS PSI input file. This file contains definition of piles and soil. The soil part defines both soil layers and soil properties (soil curves).

2.2.1 Illegally formatted soil data

If the SACS PSI file is illegally formatted a dialog box as shown below will appear.

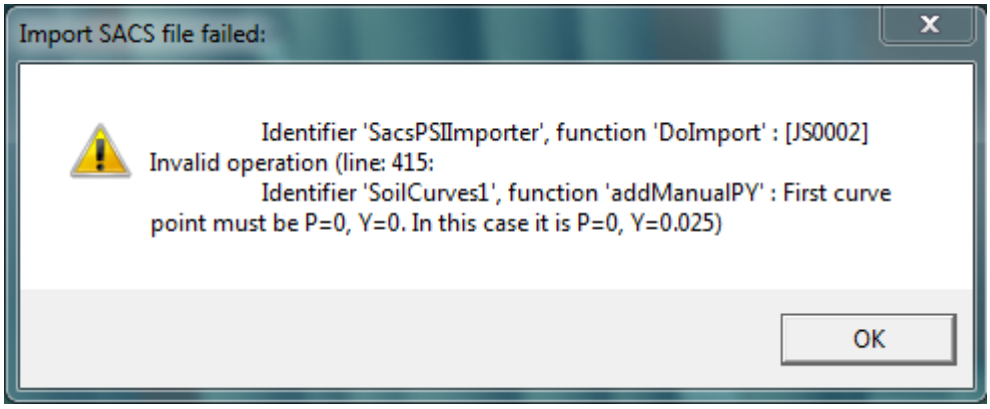


Figure 17: During import a dialog box indicating that the SACS PSI file is illegally formatted appears

An identical message appears in the Command Line tab.

```
//SacsPSIImporter.DoImport("C:/DNV/Workspaces/GeniE/jacket/psiinp.inp");
```

```
   Identifier 'SacsPSIImporter', function 'DoImport' : [JS0002] Invalid operation (line: 415:
```

```
   Identifier 'SoilCurves1', function 'addManualPY' : First curve point must be P=0, Y=0. In this case it is P=0, Y=0.025)
```

```
// Please check Messages area for 3 warnings.
```

The following message is shown appears in the Message tab.

```
Warning:
```

```
Input warnings while importing SACS PSI input file C:/DNV/Workspaces/GeniE/jacket/psiinp.inp.
```

```
 T-Z curve may have errors due to not strictly increasing displacement values in SACS input.
```

What to do:

Inspect the 'SacsPSIToGeniE.js' to identify what addManualPY-command(s) that defines (P=0, Y=0.025) as the first point.

```
SoilCurves1.addManualPY(0.61,Array(0.025 m,0.0275 m,0.0301 m,0.0352 m,0.0479 m,0.94 m),Array(0 Pa,0 Pa,0 Pa,0 Pa,0 Pa,0 Pa));
```

Locate the error in the 'psiinp.inp' file by using this information. It is, of course, possible to look for the error without going via the 'SacsPSIToGeniE.js' file.

```
SOIL          P-Y  0.000 2.500 2.770 3.260 3.990 4.790 3.990 13.94 3.990 25.38
```

```
SOIL          P-Y  3.990 94.00
```

Correct the error in the 'psiinp.inp' file.

```
SOIL          P-Y  0.000 0.000 2.770 3.260 3.990 4.790 3.990 13.94 3.990 25.38
```

```
SOIL          P-Y  3.990 94.00
```

2.2.2 Dummy bearing curve inserted

If no bearing soil curve is defined on the PSI file the mesage as shown below appear in the Message tab/ 'SacsPSIToGeniE.js' file.

```
Warning:
```

```
No supported bearing soil curve for SOL1 was found during parsing. A dummy curve is inserted.
```

The inserted dummy curve is a QZ-curve with constant pile tip resistance values equal to 0.0.

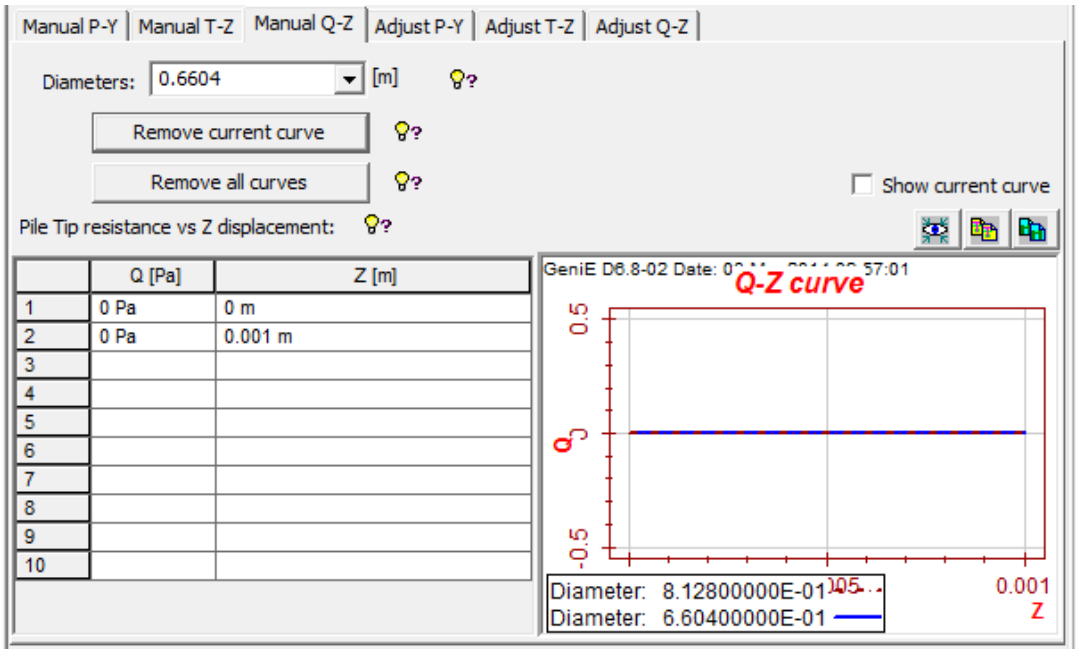


Figure 18: The dummy QZ-curve created when bearing data is missing.

2.2.3 Not supported soil curves

Currently, only manual (explicit) soil curves are supported. If the PSI file contains automatic soil curves, i.e. curves containing only basic soil properties necessary for the program to automatically generate explicit soil curves, only piles will be imported. Messages as shown below appear in the Message tab/ 'SacsPSIToGeniE.js' file.

```
Warning:
 No supported axial soil curve for SL02 was found during parsing.
Warning:
 No soil model consist of manual curves only - warn that soil could not be imported.
```

What to do:

Manually input the basic soil properties in GeniE. Program Gensod will generate all curves for all piles and conductors per user selected methods.

Alternatively, request a new PSI file containing generated manual (explicit) soil curves.

2.2.4 PSI file containing both manual curves and automatic curves

If a SACS PSI file contains manual T-Z curves and soil properties for automatic P-Y curve generation, the current importer is not able to import the PSI file directly.

What to do:

User needs to manually add dummy P-Y curves in the PSI input file, import it into GeniE, and then manually revise the P-Y curves by inputting soil type, soil data, and curve generation method for each curve.

2.2.5 Pile defined by PLSECT

A pile defined by a PLSECT card is partly supported. A message is displayed in the Message tab:

```
Warning:
```

```
Pile section (PLSECT card) data used to define outer diameter and wall thickness of pile.
Note that only pile section of type TUB is supported. Other properties than those
mentioned above is not supported.
```

In case the PLSECT cross section type is ‘TUB’, the outer diameter and wall thickness is used for defining the pile cross section in GeniE. The cross section stiffness properties data is not used.

In case the PLSECT cross section type is ‘H, the outer diameter is set to 100.0 cm and wall thickness is set to 1.0 cm for the pile cross section in GeniE. The actual cross section stiffness properties and details are skipped due to that H-profiled piles is not handled in GeniE. A message is displayed in the Message tab:

```
Warning:
Pile section (PLSECT) of type H is not supported. Tubular section data used:
Outer diameter = 100 cm and Wall thickness = 1.0 cm
```

What to do:

Manually update the relevant Section and Material properties in GeniE.

2.2.6 PSI file containing more than one soil ID

A SACS PSI file containing more than one soil ID can be converted and the soil curves with different soil ID can be assigned to piles with different pile diameters. It is possible that in such PSI file the starting layer locations for Q-Z curves are different. User needs to add dummy Q-Z curves to make Q-Z curves for different soil ID start at the same penetration in order to get the correct Q factors for all piles.

3. AFTER IMPORT

3.1 Automatic verification

If the “Model Verification” dialog appears after import it indicates that parts of the model has not been properly established. The dialog is described in detail GeniE User’s Guide “Vol 3 - Plate/Shell Structures “, chapter 3.3.7. This dialog can be activated from GeniE by using the pull-down menu ***Tools|Structure|Verify***.

Below are described some typical problems related to model problems after import.

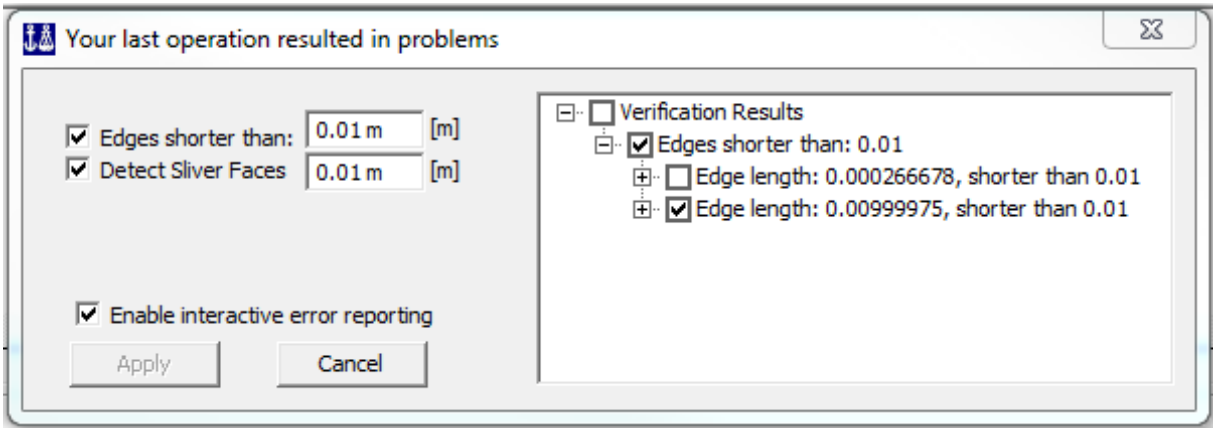


Figure 19: Model Verification dialog automatically shown after import indicating model errors

3.1.1 Short edges

When short edges are reported it might result in errors when executing the analysis. However, it is not advisable to do anything with the edges until such errors are confirmed.

In these cases Sestra may fail with message stating that

```
  *** ERROR in input on T-file (.FEM) for a 2-noded beam element
      The element length is zero or close to zero.
```

Try to divide the edge. A script command as shown below is shown in the Command Line tab.

```
Validate(Bm706.primitivePartCount == 2);
Bm706.explode(IndexedNameMask(3390));
Validate(Bm3390);
Validate(Bm3391);
```

One of the new edges, Bm3390 or Bm3391, has length close to zero and should be deleted.

This problem might be caused by member connections in SACS model. If there is a member cross over the other one without real member connection between these two members, this problem could happen during the conversion. User may need to check .CHK file, find the related SACS members, and make sure that these two members are connected at the cross over point.

Also user may need to check that the wishbone joints are at the same location as the leg joints. If they are at a slightly different location the connection will not be done correct and there can be a message “edges shorter than .01”. User will need to fix SACS input model to have wishbone joint at same location.

3.1.2 Missing properties

Missing properties originates typically from cases described in chapter 2.1.1 and 2.1.2. When such cases have been identified during import it is advisable to execute ***Tools|Structure|Verify*** from GeniE. Then the following information is shown in the model Verification dialog.

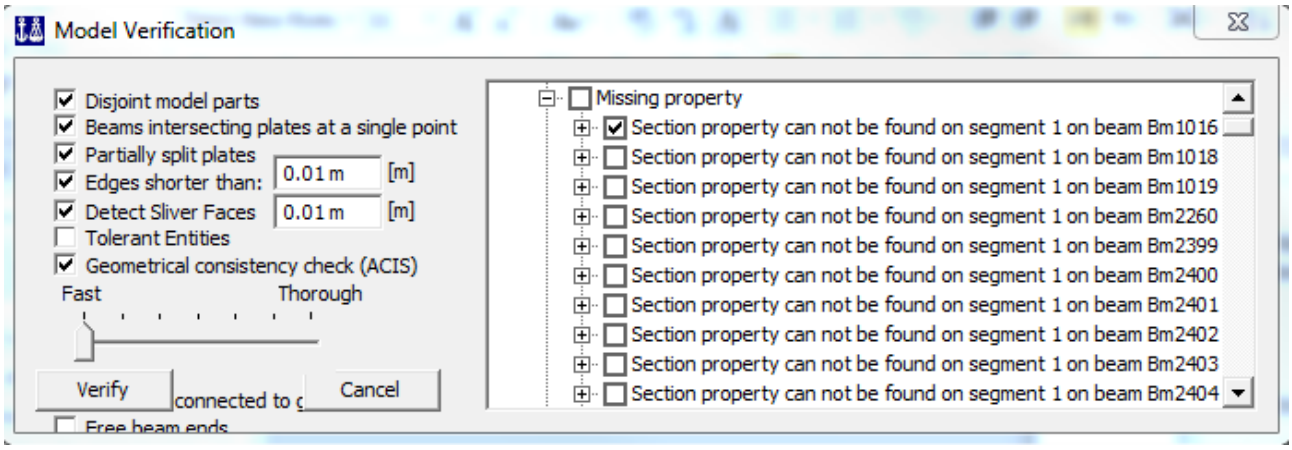


Figure 20: Model Verification dialog started from GeniE contains additional information

Checking a line will label the particular beam as show below. Beams missing cross section may also be identified in the browser. In this case <None> is written in the Section column. Columns for material or other properties may also be added to the browser and used for identification of missing properties. However, the best way of identifying what beams to be assigned to what properties is described in chapter 2.1.1.

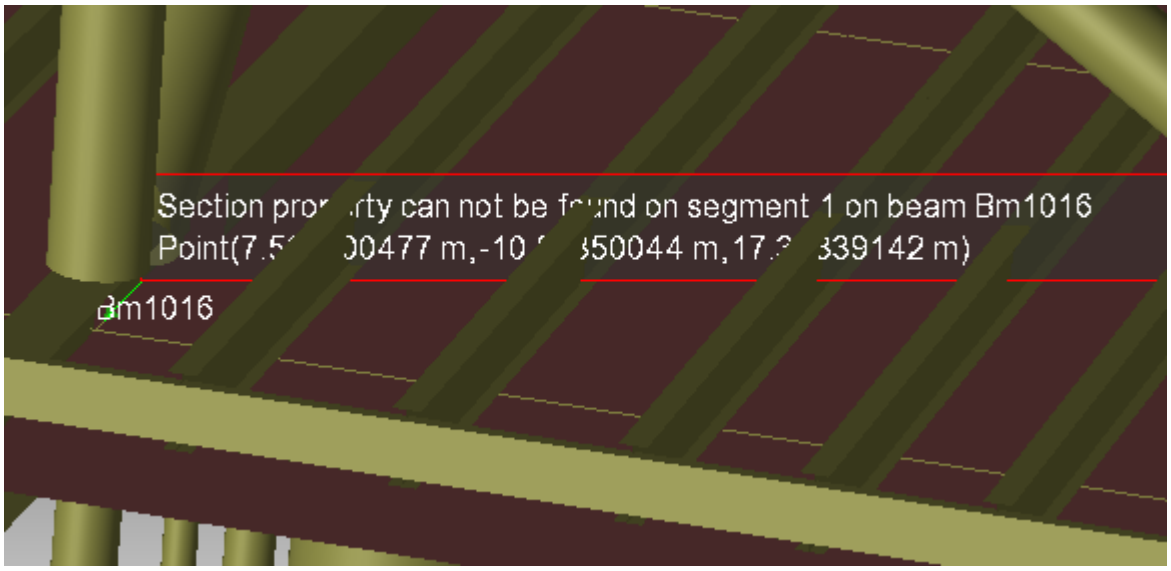


Figure 21: Labeling of problem beam by checking a line in the Model Verification dialog

3.1.3 Short edges and missing properties

As shown in the dialog in chapter 3.1 two edges are shorter than the tolerance of 0.01 m. In some cases it may be a good idea to execute ***Tools|Structure|Verify*** from GeniE. In this case it resulted in additional information; one of the short edges was missing section and material properties.

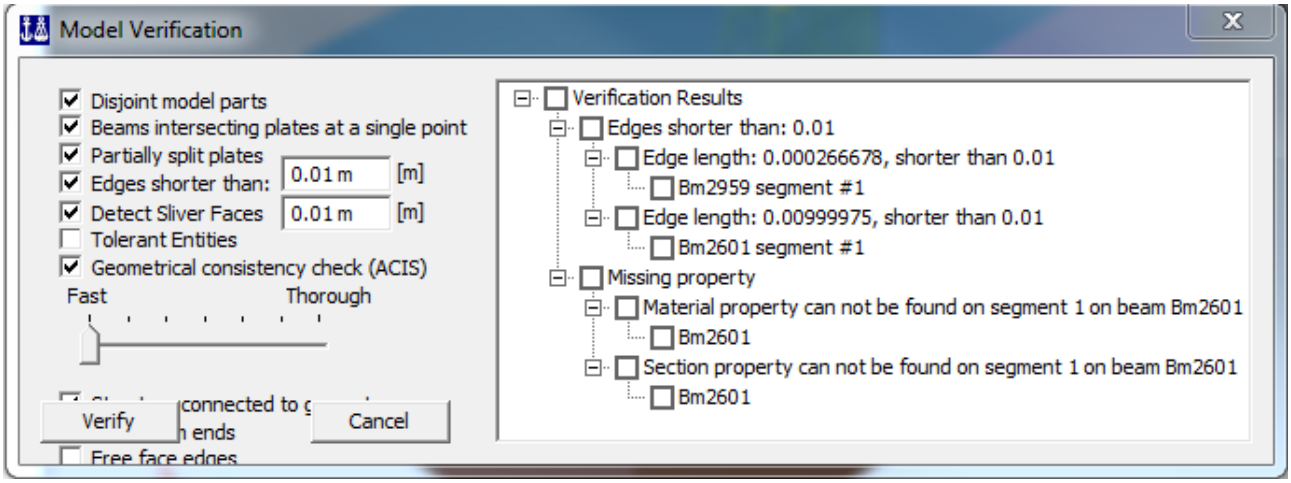


Figure 22: Combination of short edges and missing properties indicates ghost geometry

Normally it is sufficient to assign the missing properties, as shown in chapter 3.1.2. However, the combination of short edge and missing properties may be an indication of *ghost geometry*. *Ghost geometry* can neither be deleted nor repaired. In these cases it is necessary to repeat the import with a *Point tolerance* smaller than the length of the edge giving the problem.

3.2 Manual verification

By manually inspecting the result of the import, flaws in the model may be discovered.

3.2.1 Joint at unexpected position

When importing a Sacs model with 'Import named joints" option OFF (default) GeniE will insert joints where necessary. This is typically at structural joints where the Sacs input contains two or more JOINTs defined with the same coordinates. These are usually connected by wishbone elements that are converted to springs. When the structural joint is at a leg there is normally also an inner pile (overlapping beam).

Normally joints and overlapping beams are collocated along legs and conductors. If they occur at other positions, it might indicate that something is wrong and it should be investigated.

The leftmost joint shown in Figure 23 is defined because one of the caisson elements and the support beam have a common JOINT, while the other caisson element is connected to another JOINT with the same coordinate. However, no wishbone element is defined to connect these two JOINTs. To make sense the two caisson elements should have a common JOINT and the support beam a separate JOINT with a wishbone element connection. For this joint a point-point connection should probably be created before executing the analysis.

For the rightmost joint the two caisson elements are connected to separate JOINTs with identical coordinates. Also note that there is no support beam at this joint. In this case the JOINT can be deleted before executing the analysis.

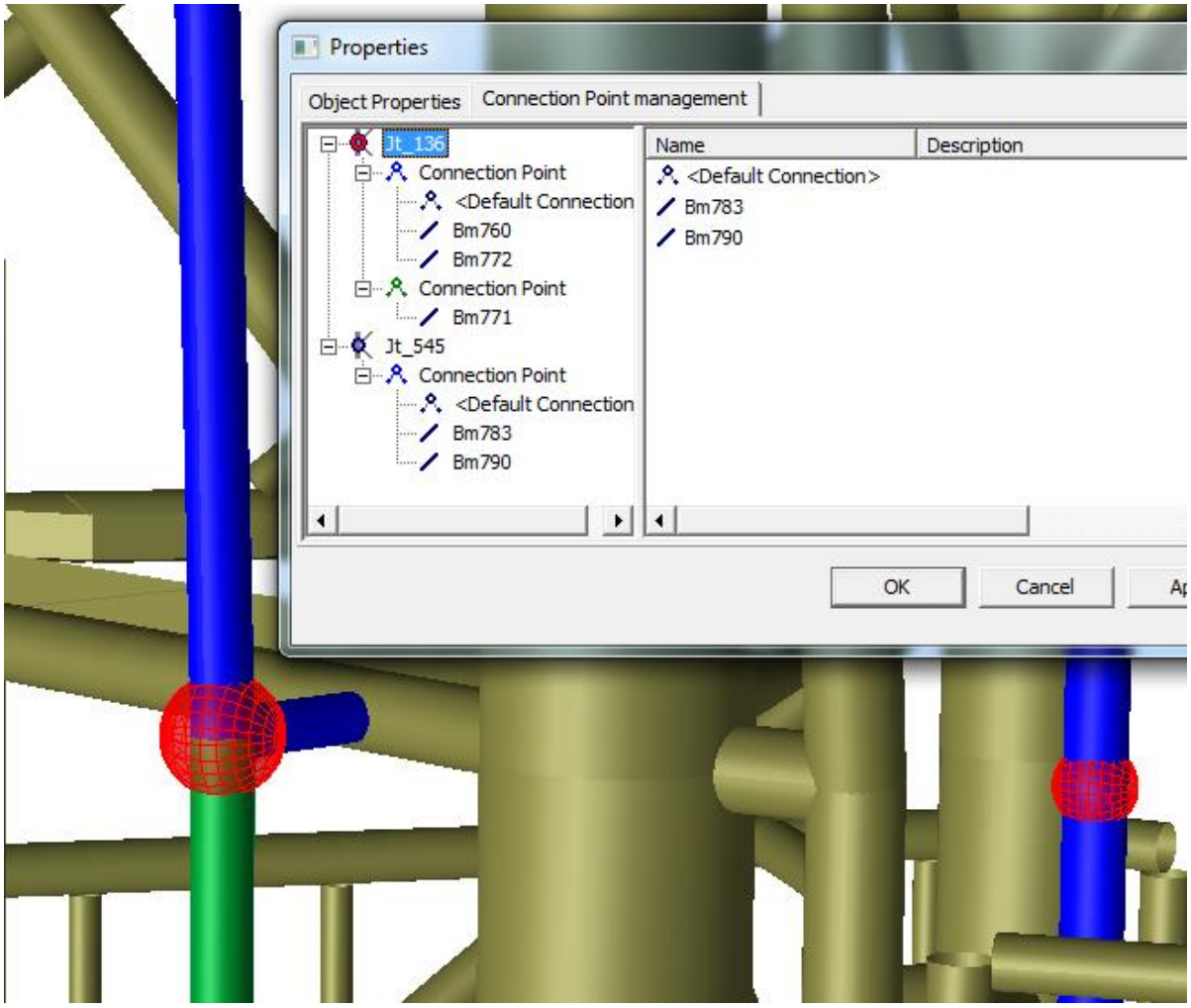


Figure 23: Joints that appear at unexpected positions may indicate errors

3.2.2 Joints at unexpected positions

A particular case where joints at unexpected positions will occur is when additional members or clamps have been modelled to replace damaged members. If the damaged members mistakenly are retained in the model, joints will be defined along with overlapping, disconnected beams. In this case the joints and beams corresponding to the damaged members should be deleted. Normally, the repair members will have different, more solid, cross section and/or material properties compared to the damaged ones. This will help the user to separate damage and repair members.

Note that Sestra will typically fail with a message as shown below in these cases.

```
      Error in factorisation of Stiffness matrix.
      Matrix is not positive definite.
      The error may be related to External Node no:    3384
      Degree of freedom no:   1
      Coordinates of the node:  3.000E+00  1.459E+01  2.300E+00
      Connected to External Element no:    5519
      Connected to External Element no:    5834
```

What to do:

Detect problematic beams by graphical inspection. Two alternative methods are described. Note that visualisations of joints should be turned on.

1. Model has been imported with the “Detect disconnected inner beam” checked (default).

   In the Properties->BeamTypes folder right-click on the “Inner Beam Disconnected” property and select “Color Code Property”. Inspect all disconnected inner beams and evaluate their position, see Figure 24.

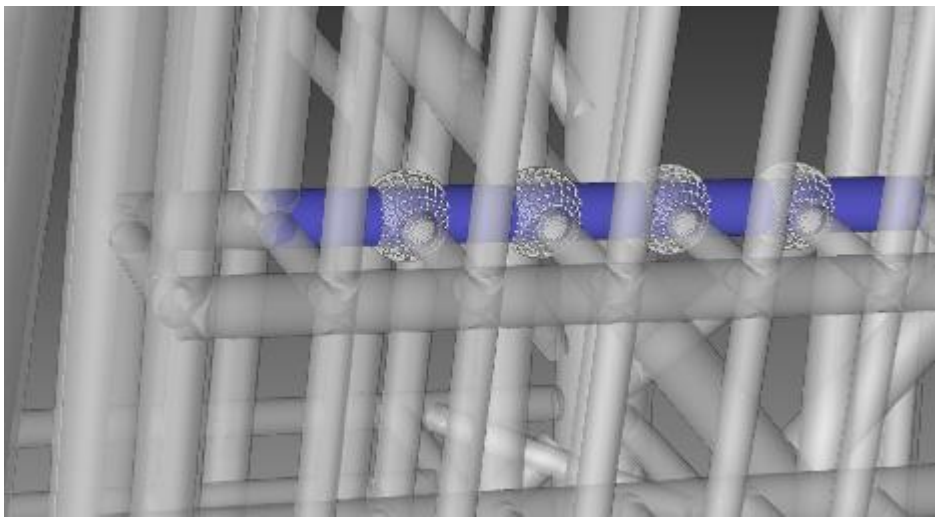


Figure 24: Color coding of Inner beam disconnected property

2. Model has been imported with the “Detect disconnected inner beam” unchecked.

   Sort on Description in the Structure folder and select all beams with description “Overlapping Straight Beam”. Inspect all highlighted overlapping straight beams and evaluate their position, see Figure 25.

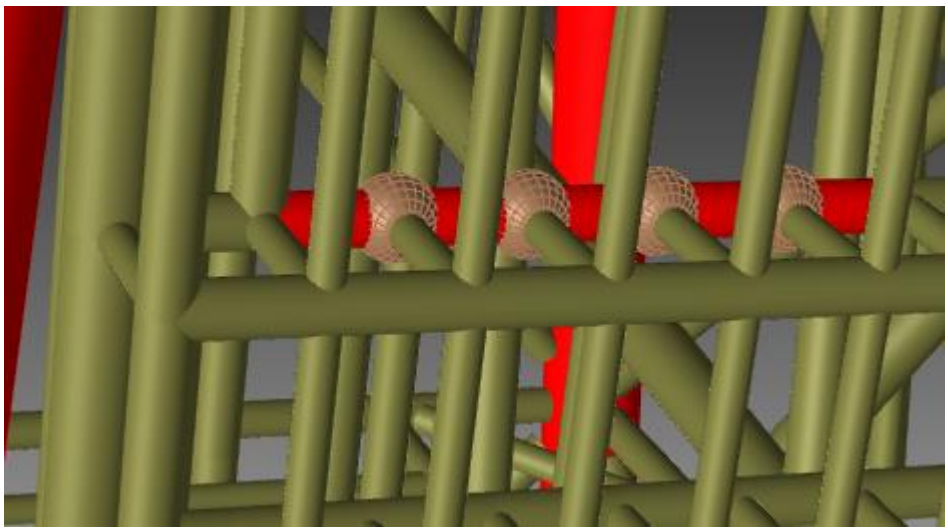


Figure 25: Highlighted overlapping straight beams

The user should evaluate which beams that may be deleted. Normally the joints may also be deleted.

3.2.3 Cross sections representing dented tubular members

Dented tubular member cross sections are converted and imported as tubular sections without any dent or grout information. Check for Warning message on the LOG file if dented tubular sections are present in the imported model, typically:

```
Warning: SECT DTB1    is a dented tubular, all dent info will be skipped
```

The user must manually perform modifications/updates regarding how to treat beams with dented tubulars with respect to structure analysis and post-processing.

3.3 Manually re-engineer converted conical members (V7.0 and older)

Note that from GeniE version 7.1 the procedure described here is no longer necessary. Conical members are automatically created in the converted GeniE model.

During import each cone (CON) section is converted to a pipe section with average diameter. The user has to manually re-engineer these pipe segments back to conical segments before executing the analysis and running code checks.

Note that from GeniE V7.1 most conical segments are correctly imported as long as the "Import mesh into analysis" option is checked (default). Manual actions are no longer required. Due to a limitation in GeniE, segments that are not internal on the beam cannot be assigned a cone section. These segments have to be assigned a pipe section, and preferably the one with average diameter assigned during import.

Note that warning messages are written to the log file with respect to GRUP definitions where a conical transition is assigned to the first or last member segment. This also applies to member GRUPs which consist of one cone only. Typical warnings:

```
Warning: Non-supported use of CON for GRUP D01. Cone defined at first segment or whole member.
Warning: Non-supported use of CON for GRUP D02. Cone defined at last member segment.
```

What to do:

The CHK file contains a "Cross section table" that lists the original name and type for each section. An example of the table is shown in Figure 26. To re-engineer converted conical members do the following:

1. Identify and select the corresponding sections in the Properties Sections folder (Figure 27), RMB and select Color Code Property.
2. Inspect the color coded pipe segments, shown in Figure 28. Perform steps 3 and 4 for each color coded segment.
3. Apply a cone section to the segment as shown in Figure 29. Note that a cone section must be created if it does not already exist. Also note that beam selection must be in "Filter Segment" mode.
4. Apply correct thickness to the cone segments. Plate thickness according to the tubular wall thickness shown in Figure 27 must be created if it does not already exist.

```
Cross section table
       Name                 Type
-----------------------------------------

     CONCH12                  Cone Section
     CONCH21                  Cone Section
     CONCH22                  Cone Section
     CONCH31                  Cone Section
     CONCH32                  Cone Section
     CONCH41                  Cone Section
     CONCH42                  Cone Section
```

Figure 26: Cross section table showing original Cone sections

Name	Description	Type	Diameter [m]	Thickness [m]	Area [m^2]
CONCH12	Pipe Section, d=1.042, t=0.0115	Pipe Section	1.042	0.0115	0.0372302
CONCH21	Pipe Section, d=1.142, t=0.0175	Pipe Section	1.142	0.0175	0.0618226
CONCH22	Pipe Section, d=1.092, t=0.0145	Pipe Section	1.092	0.0145	0.0490835
CONCH31	Pipe Section, d=1.142, t=0.0175	Pipe Section	1.142	0.0175	0.0618226
CONCH32	Pipe Section, d=1.092, t=0.0145	Pipe Section	1.092	0.0145	0.0490835
CONCH41	Pipe Section, d=1.092, t=0.0175	Pipe Section	1.092	0.0175	0.0590737
CONCH42	Pipe Section, d=1.042, t=0.0115	Pipe Section	1.042	0.0115	0.0372302

Figure 27: Corresponding pipe sections shown in the Sections folder

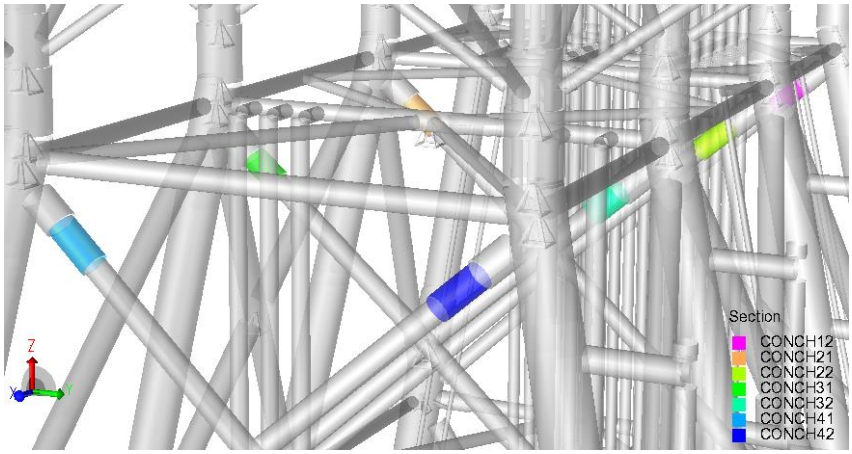


Figure 28: Color coded selected pipe sections

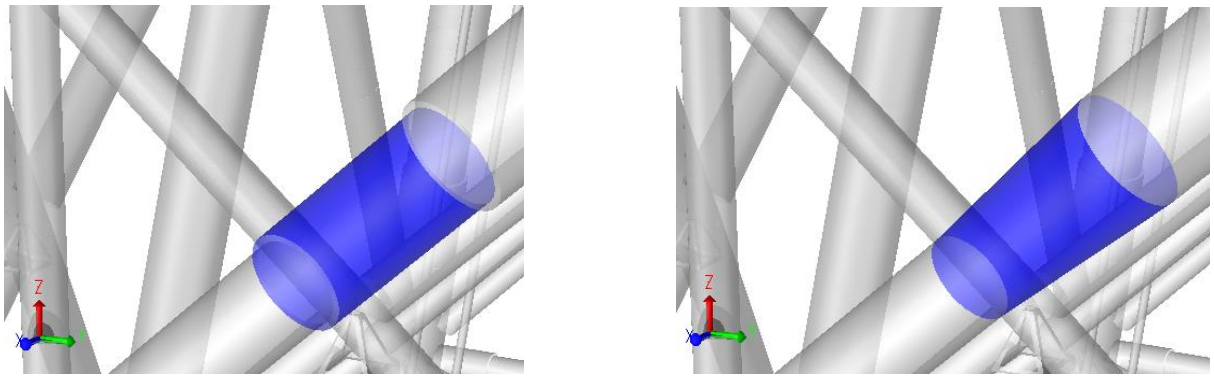


Figure 29: Assign a cone section to relevant segments, and then apply correct thickness

3.4 Prepare for joint code check

If a joint code check is intended, the model should be imported with the "Import named joints" option checked. In this case Joint Cans should be assigned to joints where necessary. The procedure is as follows:

1. Select relevant joints and execute the "Assign Joint Can" command

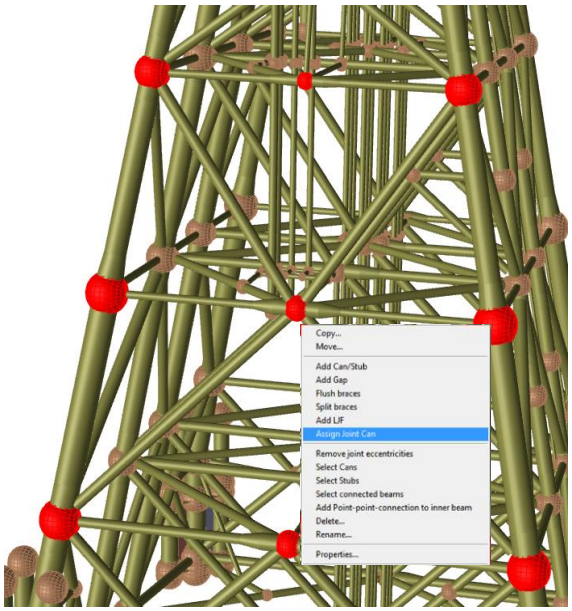


Figure 30: Select relevant joints and execute the "Assign Joint Can" command

2. Inspect the result of "Assign Joint Can" command

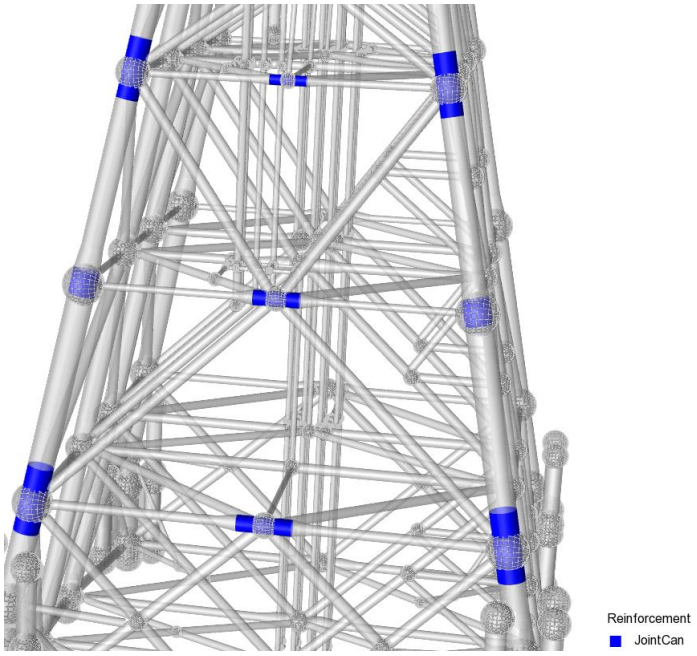


Figure 31: Result of "Assign Joint Can" command

For a more detailed explanation consult the GeniE user manual Vol 1, chapter 3.8.2.2.

3.5 Manually added "Adjust Q-Z" data for pile soil analysis (V7.2 and older)

SACS allows the user to define the pile end bearing area on the PLGRUP line. This information is not read by the SACS importer. Moreover, as Splice will use the entire cross section area as the end bearing internally, "Adjust Q-Z" data must be manually defined according to the following rules.

1. If a pile group has several segments, only the end bearing area on the last segment is used.
2. The compression factor "Cmp-Fact" is defined as follows
    a. If end bearing area (A) is defined on the PLGRUP line

Cmp-Fact = A/"entire pile cross section"

b. If no end bearing area is defined on the PLGRUP line

Cmp-Fact = "pile steel area"/"entire pile cross section"

Example

In the SACS PSI file the following end bearing areas were defined

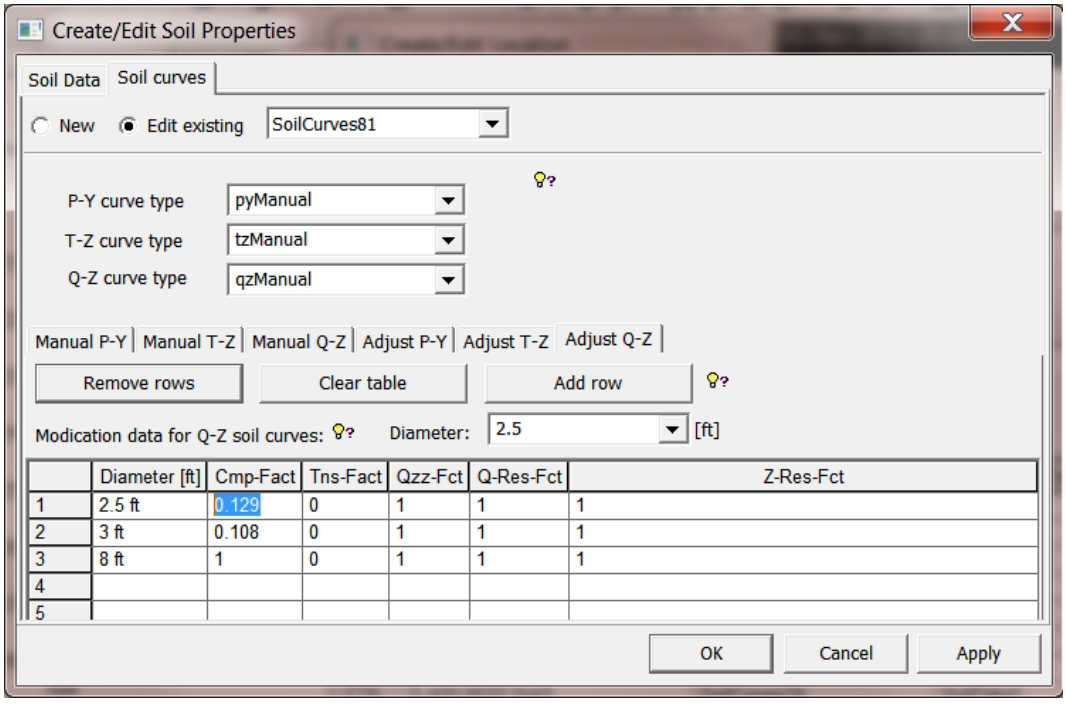
```
PLGRUP
PLGRUP S1A            96.  2.25   29.  11.6  50.0   176.0              1.0
PLGRUP S1A            96.  2.50   29.  11.6  50.0    10.0              1.0 50.26
PLGRUP CN1            36.   1.0   29.  11.6  36.0   150.0              1.0 .7632
PLGRUP CN2            30.   1.0   29.  11.6  36.0   150.0              1.0 .6323
```

Cmp-Fact for S1A is (96 in equals 8 ft) = $50.26/(4^2*3.14) = 1.0$

Cmp-Fact for CN1 is (36 in equals 3 ft) = $0.7632/(1.5^2*3.14) = 0.108$

Cmp-Fact for CN2 is (30 in equals 2.5 ft) = $0.6323/(1.25^2*3.14) = 0.129$

The factors applied in the SoilCurve dialog. Note that the factors should only need to be defined for the layers containing the pile head.

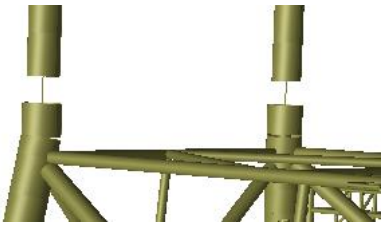


3.6 Manually re-engineer beams with DefaultCone section (V7.3 and older)

When the "Use Domain Model Import" option is checked sections named "DefaultCone" may be created and assigned to beams.

Name	Description	Type	Diameter [m]
A36X170	I Section, h=0.91948, w=1.2192, wt=0.017272, ft=0.02794	I Section	
A36X441	I Section, h=0.98806, w=1.2192, wt=0.034544, ft=0.061976	I Section	
DefaultCone	Cone Section, tweight=0 ,manual=true	Cone Section	

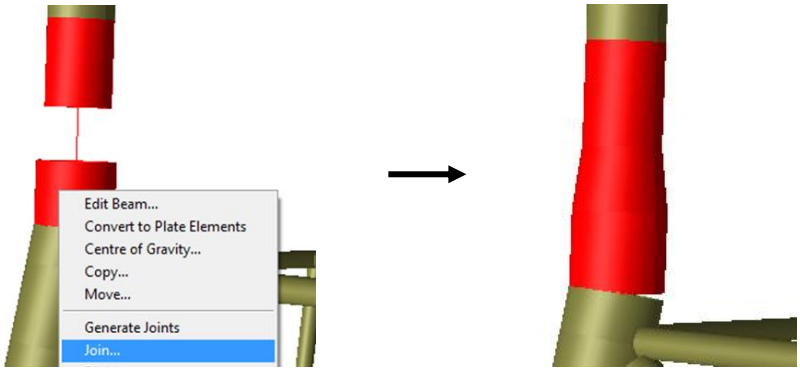
If this section is assigned to a one-segmented beam or to a segment at one of the beam ends, the beam will have no visible cross section.



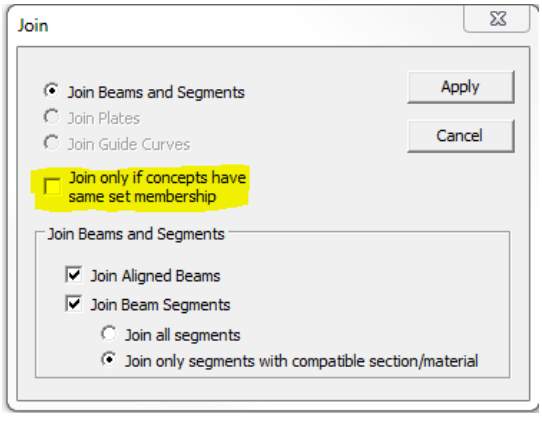
Moreover, Sestra will fail.

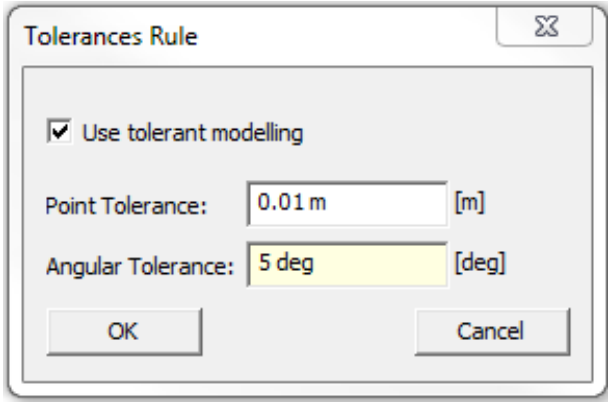
```
*** ERROR in input on T-file ***
    for superelement type     1 on level  1
    Check of T-file (.FEM) revealed the following errors:
  * Geometry data is missing for element   1871
    (the reference number in GELREF1 is zero)
```

The problem is fixed by selecting the adjacent beams, right-click and select “Join…”



Note that the “Join only if concepts have same set membership” option must be unchecked prior to pushing Apply. If the beams are not joined it may be necessary to **temporarily** increase the Angular Tolerance.

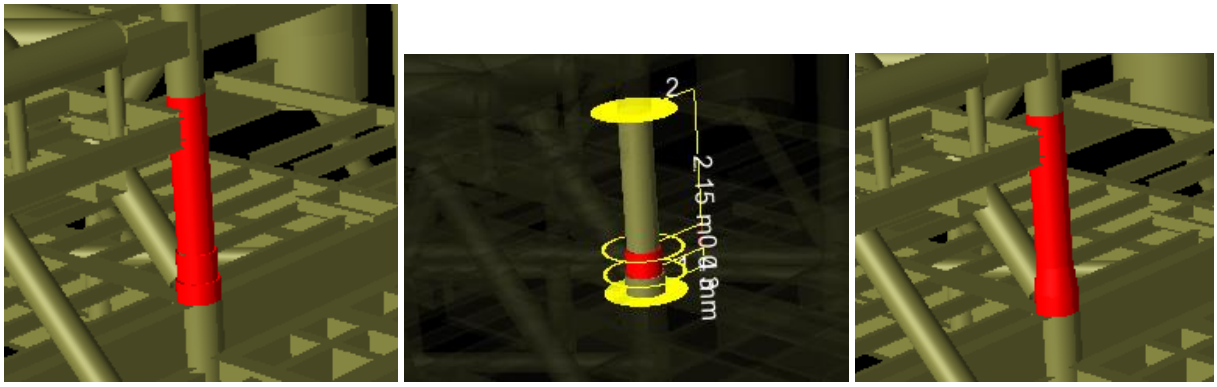




Note that the GeniE v7.3 and older an averaged pipe section is assigned to such beams.

The problem is fixed by selecting the adjacent beams, right-click and select “Join…”

Select the middle segment and define property as DefaultCone.



3.7 Manually Delete Overlapped Beams under the mudline

Note that from GeniE version 7.4 the procedure described here can be used. Under Structure-Utilities-Sets-Regular Sets a Set “BELOWMDL” will be created with all members below the mudline. These members can be checked if any are specified as overlapped beams. If specified as an overlapped beam they will need to be deleted.

4. DURING ANALYSIS

4.1 Wave Load Analysis fails

If Wave Load Analysis fails the Wajac.LIS file should be inspected. Look for error messages. Read the Wajac User Manual to get additional information. In particular see chapter 4.4 - Warnings and Error Messages.

4.1.1 Wave height problem

Wave Load Analysis may fail with the error message

```
** WARNING ***: WAVE HEIGHT   24.00    (IWAVE=    2) EXCCEDS HEIGHT OF BREAKING WAVE
22.16
     WHICH IS  0.142 TIMES WAVE LENGTH (DEEP WATER BREAKING LIMIT, AIRY THEORY)
   2 STOKES 5TH   10.00            24.00       182.7      0.6789        0.1314
138.7       114.7       4.233       5.767
```

This indicates a problem with wave height exceeding the breaking wave limitations enforced by Wajac.

What to do:

Ensure that Wajac version is 6.3-02 or newer is used and if necessary install a new version of Wajac. If this does not solve the problem, check the “Ignore breaking wave height limit” option in the “Special options” tab of the “Edit Wave Load Run” dialog and rerun the analysis.

Note that some wave models are more permissive when parameters go outside legal range. Airy will always work and Stokes 5th order is more likely to produce results than Stream Function.

Note also that results should be interpreted with care when using a wave theory outside its validity range.

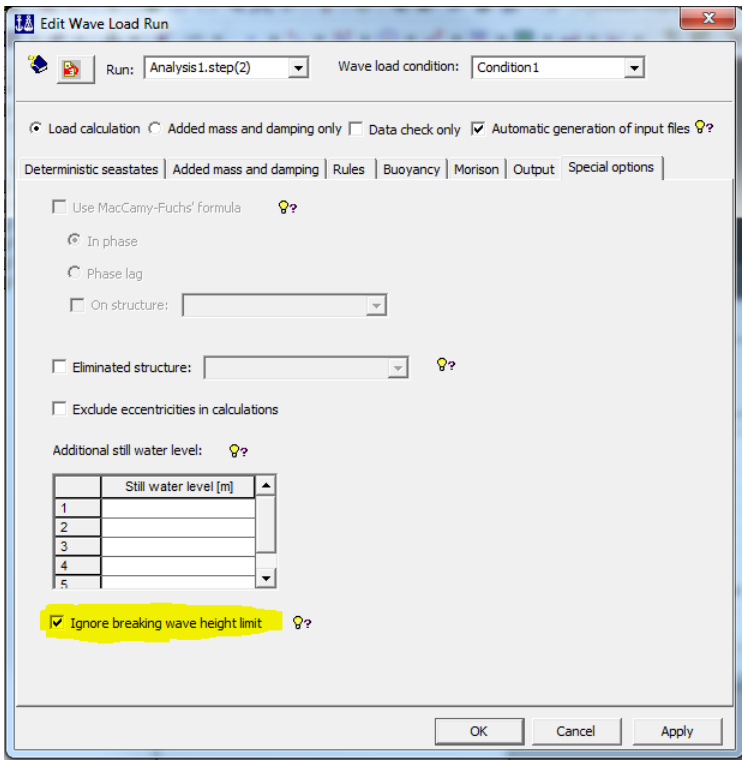


Figure 32: Checking the "Ignore breaking wave height limit" will solve the problem

4.1.2 Not able to calculate half wave period

Wave Load Analysis may fail with the error message

```
*** WARNING FROM SEAD83
     ERROR FLAG =    -7

         WAVE INDEX   : IWAVE =          12
         WAVE PERIOD : T      =  2.210
         WAVE HEIGTH : H      =  1.930
         THEORY FLAG : ITHEO=           6

     NOT ABLE TO CALCULATE HALF WAVE PERIOD
     ERROR RETURN FROM SUBROUTINE T2CT2T, IER = -3
     CHECK YOUR INPUT PARAMETERS
```

This indicates a problem with wave height that is too high compared to relevant water depth and period. It exceeds the range of the Stream Function.

What to do:

Change wave theory to Airy or lower the wave height through Condition in Location.

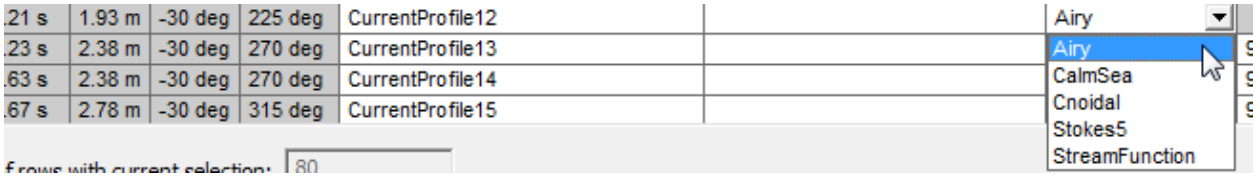


Figure 33: The wave components table of the "Edit Wave Load Condition" dialog

Run analysis again.

4.1.3 Non-flooded area smaller than flooded area (V6.9 and older)

Please note that in SACS any kind of override displacement areas can be used in buoyancy calculations, while in Wajac the defined non-flooded areas cannot be smaller than the flooded areas.

If the SACS input file contains GRPOV/MEMOV lines with override displacement (non-flooded) areas smaller than steel cross section (flooded) areas, Wajac will fail with a message similar to:

```
THE NON-FLOODED BUOYANCY AREA 0.6458E-04 IS SMALLER THAN THE FLOODED BUOYANCY AREA  4.602
```

What to do:

In order to fix the problem the user may redefine the flooded area in GeniE and make it the same as the non-flooded area.

Example:

Member is flooded with small override displacement area in SACS input file. After it was imported into GeniE, the buoyancy area like in Figure 34 was defined and Wajac reported the error message shown above.

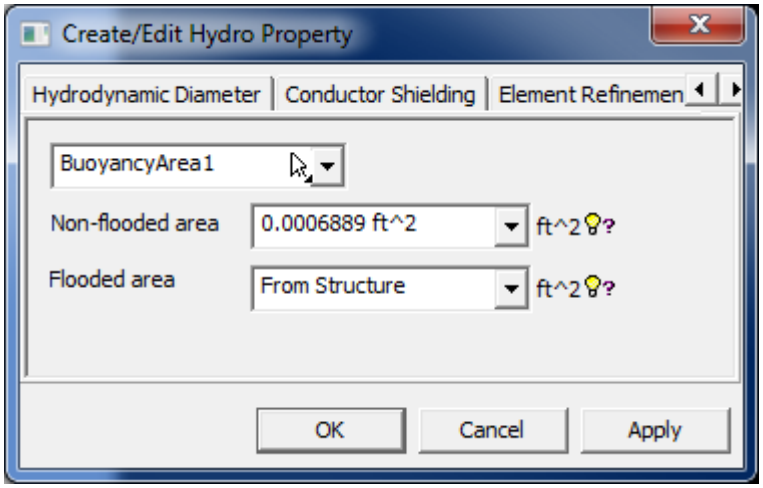


Figure 34: Buoyancy area definition resulting in Wajac error

When revising the buoyancy areas like shown in Figure 35, Wajac worked fine with the correct calculated buoyancy and steel weight.

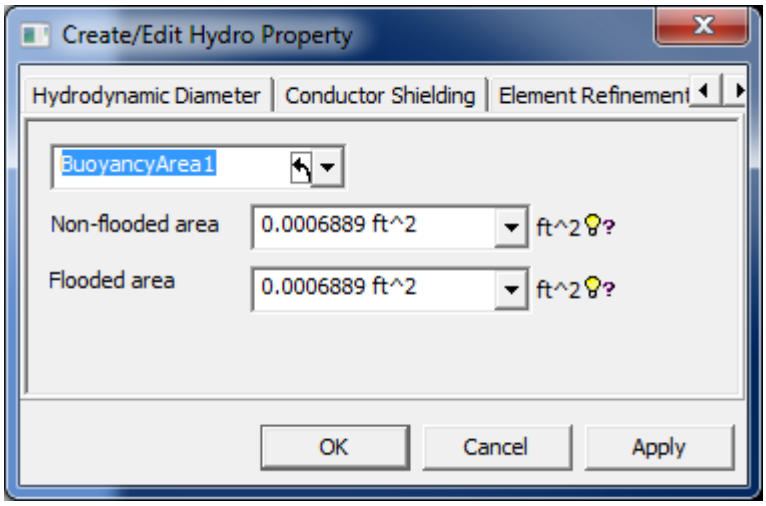


Figure 35: Revised buoyancy area definition

Note that a non-flooded member with the same small override displacement area will behave exactly same as the flooded member.

From V7.0

From GeniE V7.0, if the SACS input file contains GRPOV/MEMOV lines with override displacement area less than 1.0 mm^2, the override cross section area will be set to 95% of the displacement area.

From version GeniE V7.3 the override cross section area is set equal to the displacement area in these cases. A warning will be issued in the LOG file/Message tab More extensive information is found in the CHK file.

What to do:

In this case no manual actions are necessary.

4.2 Linear Structural Analysis fails

If Linear Structural Analysis fails the Sestra.LIS file should be inspected. Look for error messages. Read the Sestra User Manual to get additional information. In particular see chapter 4.4 - Warnings and Error Messages.

4.2.1 Illegal connection points

Linear Structural Analysis fails with the error message

```
      Error in factorisation of Stiffness matrix.
      Matrix is not positive definite.
      The error may be related to External Node no:     371
      Degree of freedom no:   1
      Coordinates of the node: -4.572E+00 -7.087E+00  4.270E+00
      Connected to External Element no:     496
      Connected to External Element no:     497
      Connected to External Element no:     614
```

This is an indication of not well conditioned boundary conditions which can have several reasons. In this case the problem is inner beams disconnected classification resulting in wrong connection points. The model must be corrected in several steps.

What to do:

Perform a ‘Locate FE…’ in ‘Mesh - All’ view combined with selecting and viewing all beams with ‘Inner Beam Disconnected’ in ‘Modelling - Structure’ view may show that a particular beam has got an inner beams disconnected classification resulting in the Sestra error.

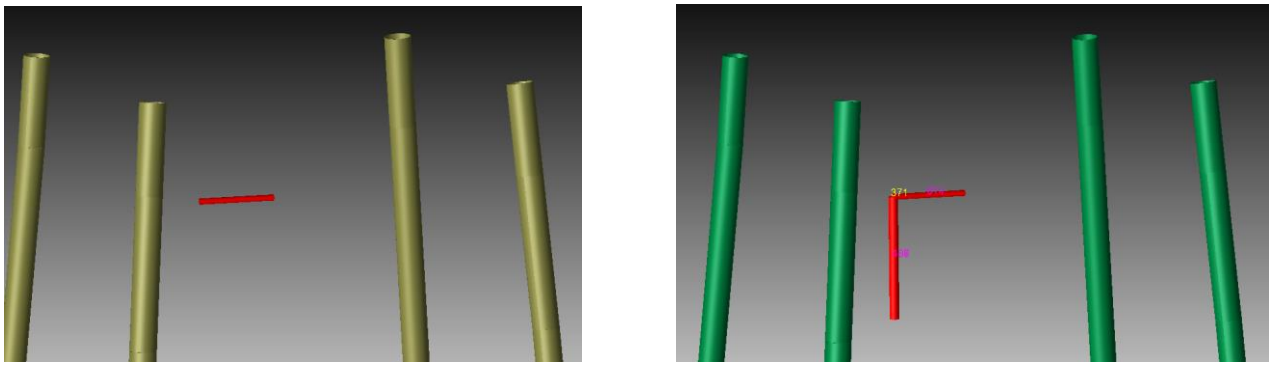


Figure 36: Graphical view of beams with Inner Beam Disconnected property and information from Sestra.LIS may help to understand the problem

Select the joint that corresponds to the node and select ‘Properties…’ from the RMB menu. As shown is the number of point connections wrong. Merge the two connection points into one.

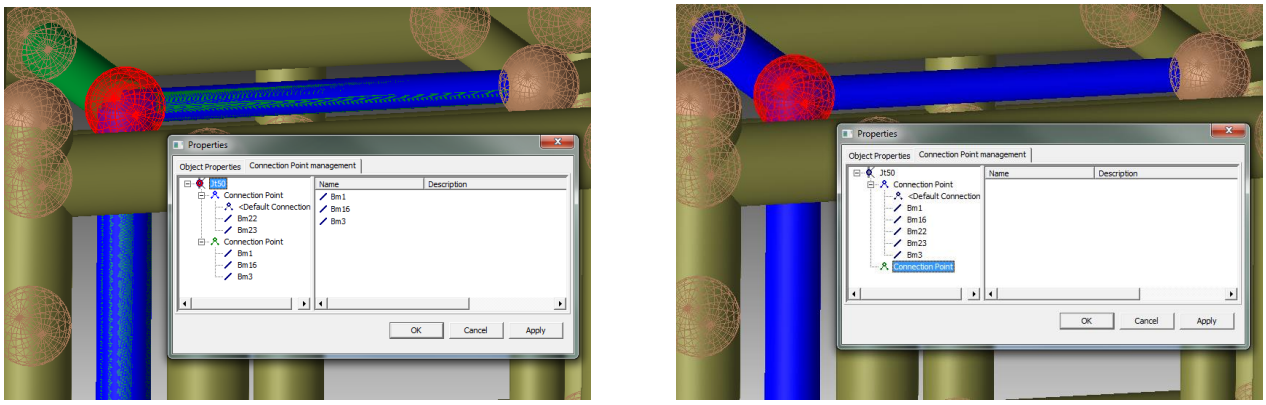


Figure 37: Merge the two connection points into one may resolve the Sestra error

It may be necessary to repeat this last operation several times to remove all similar types of Sestra errors.

4.2.2 Inner piles not properly connected

Linear Structural Analysis fails with the error message

```
Error in factorisation of Stiffness matrix.
Matrix is not positive definite.
The error may be related to External Node no:    1030
Degree of freedom no:    6
Coordinates of the node: -3.143E+01  1.269E+01 -1.450E+01
Connected to External Element no:    1033
Connected to External Element no:    3547
```

This is an indication of not well conditioned boundary conditions which can have several reasons. In this case the problem is that inner piles are not properly connected to legs extending from deck structure. The reason is the “Inner Beam Disconnected” property of the inner beam at the top of the leg above the joint. This result in two nodes, 4 and 1033, being created at the upper end of this beam which in turn leads to that the inner beam is not fixed against rotation.

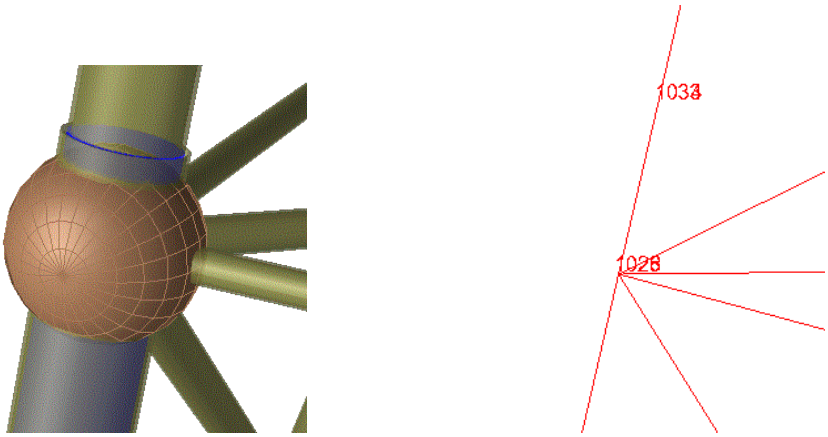


Figure 38: The Inner Beam Property of beam above the joint results in bad connectivity

What to do:

In this case the solution is to add a joint on top of the leg and make sure that the correct beams are connected. After having inserted the joint, “right click” it and select “Properties”. Then you have to move the connected beams to the correct connection point. This may be necessary to do for several legs.

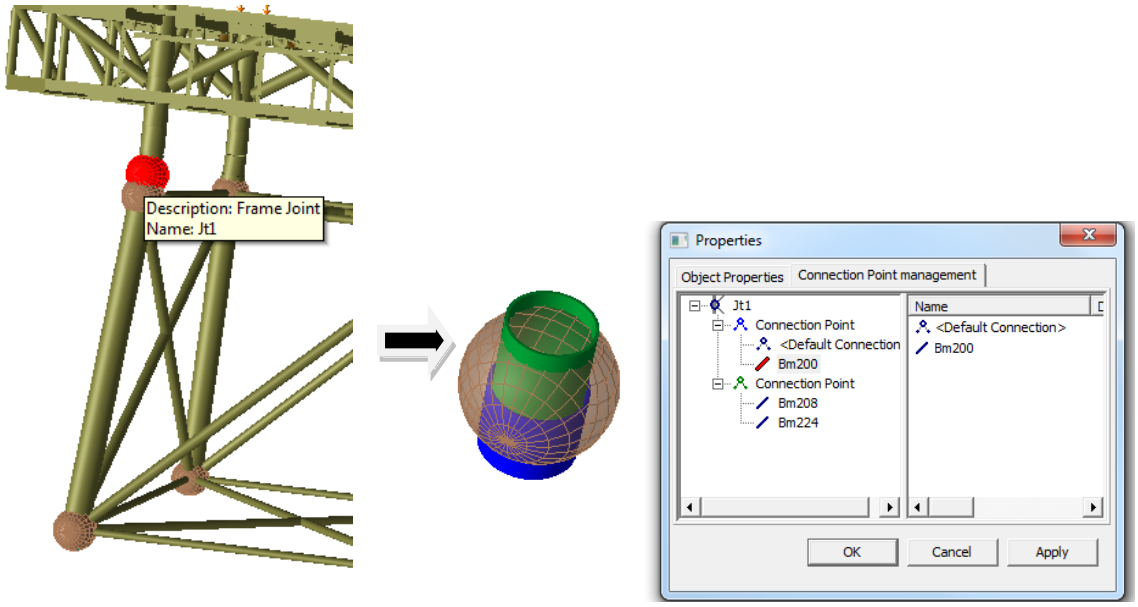


Figure 39: Add a joint on top of the leg and manually modify the connectivity and ensure correct connectivity by using the Connection Point management.

4.2.3 Missing Inner Beam Disconnected property

Linear Structural Analysis fails with the error messages

```
*** ERROR *** Element     3800 is connected twice to the same node     1160
*** ERROR *** Element     3804 is connected twice to the same node     1151
*** ERROR *** Element     3805 is connected twice to the same node     1189
*** ERROR *** Element     3809 is connected twice to the same node     1188
*** ERROR *** Element     3810 is connected twice to the same node     1226
*** ERROR *** Element     3814 is connected twice to the same node     1222
*** ERROR *** Element     3815 is connected twice to the same node     1355
*** ERROR *** Element     3819 is connected twice to the same node     1320
```

This problem occurs after re-meshing because the beam property type (BmType1; Inner Beam Disconnected) has not been applied to the “Inner beams” at both sides of the connection points. In the example shown in Figure 40 the beams representing the conductors have this property type assigned above the conductor area level only (blue color coding).

What to do:

Assign the property type to the conductors below the conductor support level as well. Color code the beam property and look for similar configurations if the meshing error still occur.

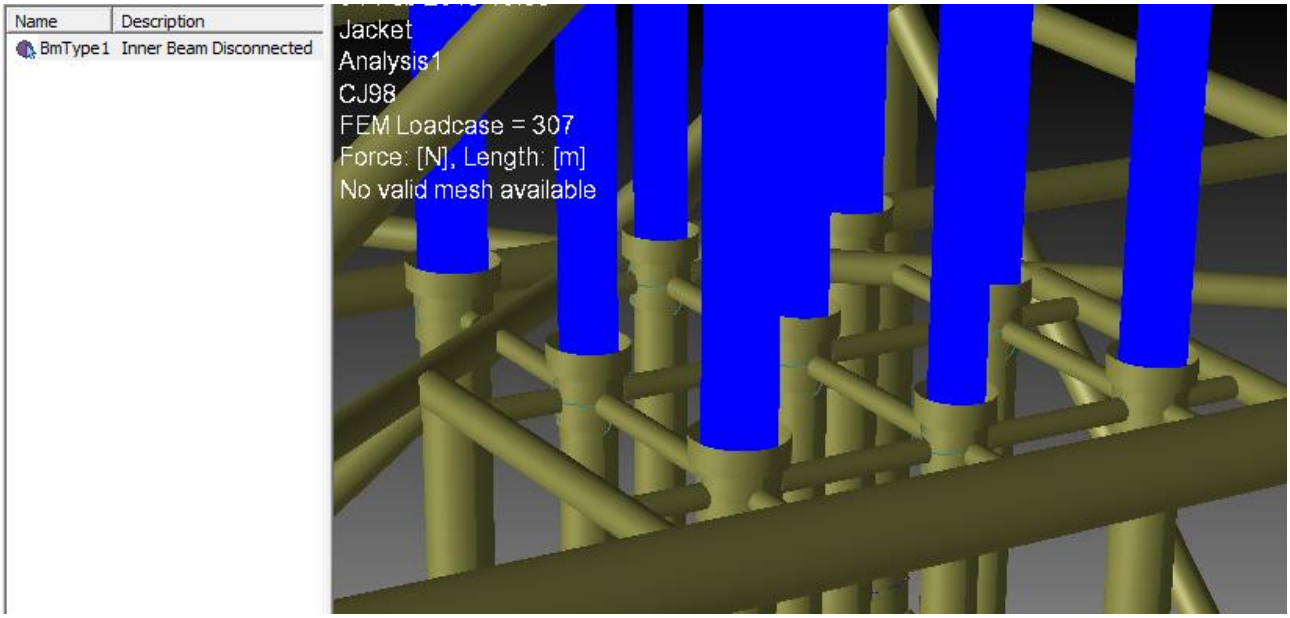


Figure 40: Color code the beam property and look for similar configurations if the error still occurs

4.2.4 Element shape problem

Linear Structural Analysis fails with the error message

```
*** ERROR *** The run terminated for element number     279
                  Element type =  25 (FTRS, 3-n shell)
```

This indicates a serious problem with the element shape, probably enforced by some geometrical irregularity.

What to do:

Performing a ‘Locate FE…’ in ‘Mesh - All’ view shows the illegally shaped shell element.

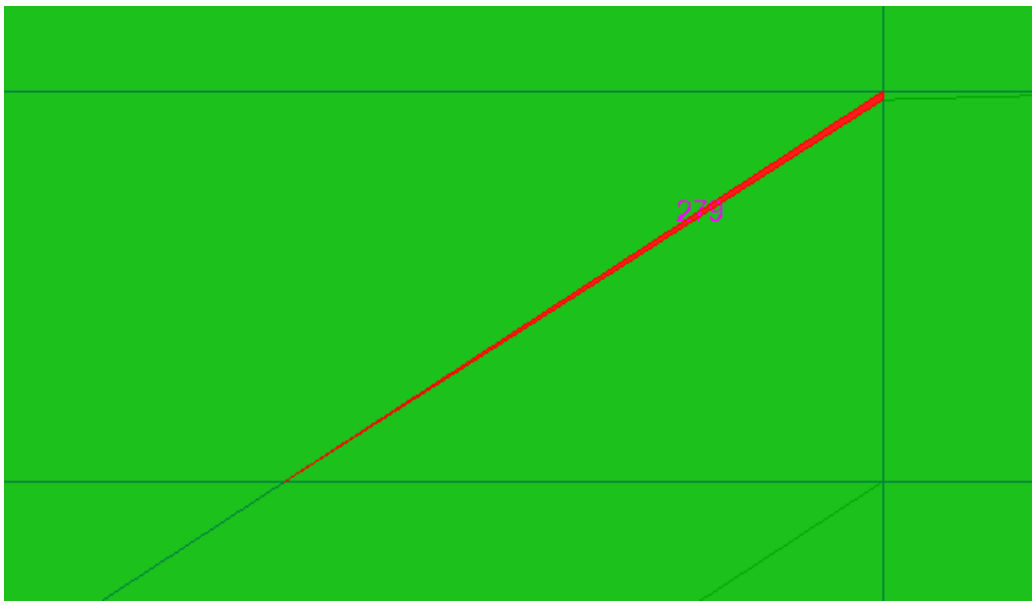


Figure 41: Graphical view of illegally shaped element

Next step is to inspect the geometry in the region of the element. Figure 42 shows the small edge causing the illegally shaped element. The reason for this edge is the inaccurate positioned beams. A manual healing of the geometry is needed. Start with deleting the joint.

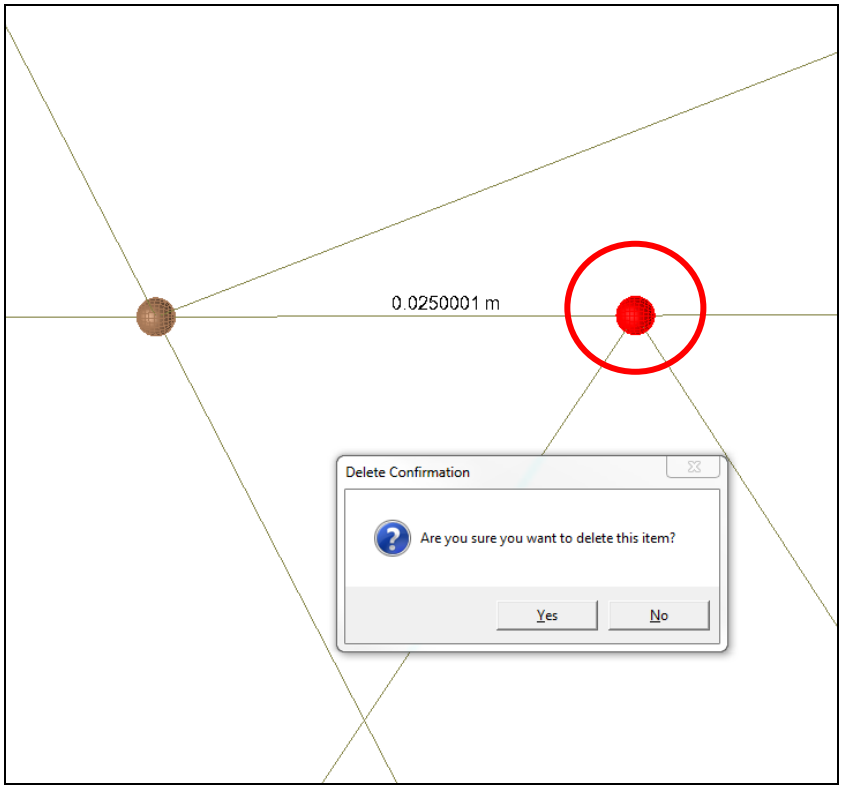


Figure 42: Inspecting the geometry shows a small edge due to inaccurate positioned beams. A manual healing of the geometry is needed. Start with deleting the joint.

Continue with moving the ends of the two inaccurate positioned beams. Finally join the two co-linear beams.

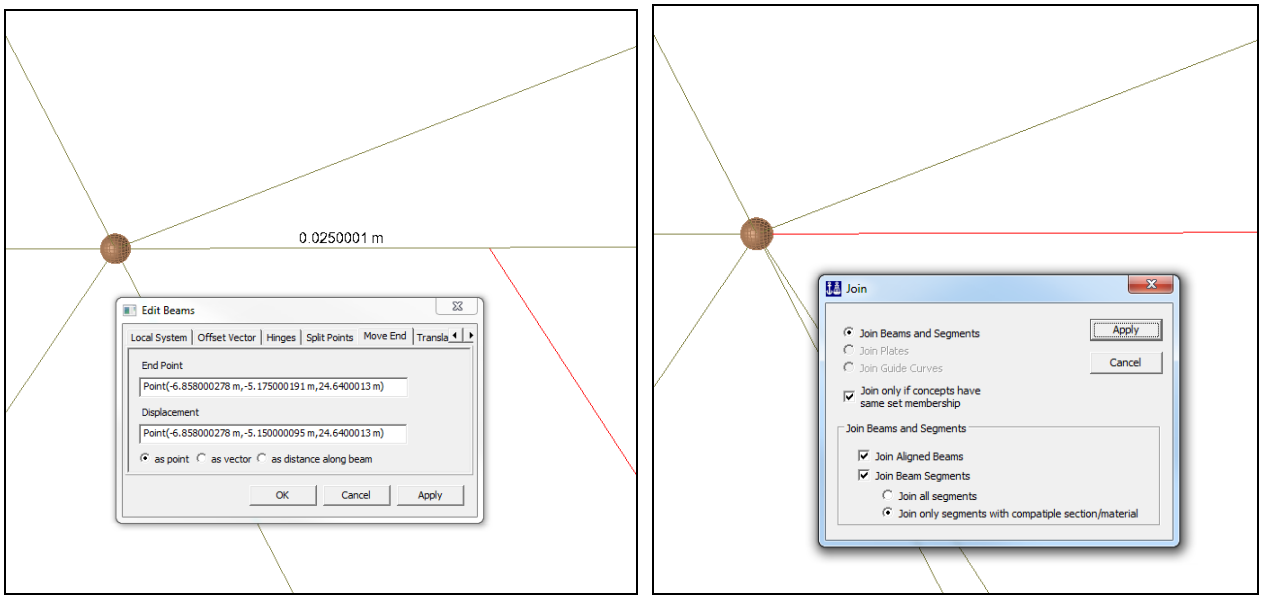


Figure 43: Continue with moving ends of inaccurate positioned beams and join the co-linear beams.

After re-meshing the illegally shaped element is removed as shown in Figure 44.

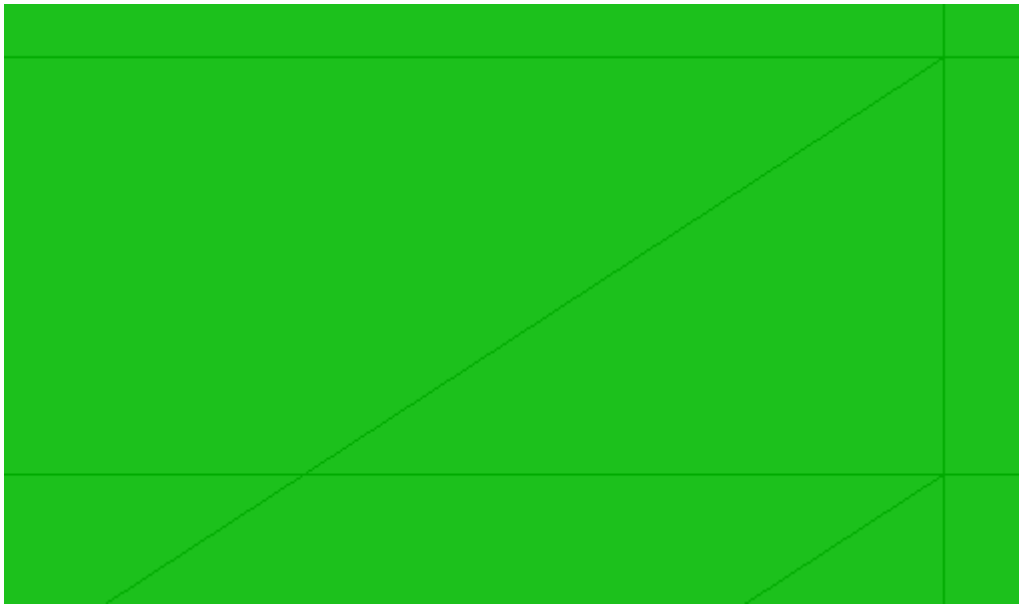


Figure 44: After re-meshing the illegally shaped element is removed

Executing the analysis after manually healing the geometry may result in a similar error message from Sestra pointing to another element. Thus, these operations may have to be repeated several times until the analysis executes successfully.

4.2.5 Hinge causing element stiffness problem

Linear Structural Analysis fails with the error message

```
 *** ERROR *** Hinges caused error for element stiffnes.
 *** ERROR *** for element no.     113  2-n beam (BEAS)
```

This indicates that one or more beams has hinges with definition that violates stiffness criteria.

What to do:

Performing a ‘Locate FE…’ in ‘Mesh - All’ view shows the beam element causing the problem. Switch to Structure view and select “Edit Beam” option by RMB on the corresponding beam. By selecting the Hinges tab the hinges are shown, see Figure 45.

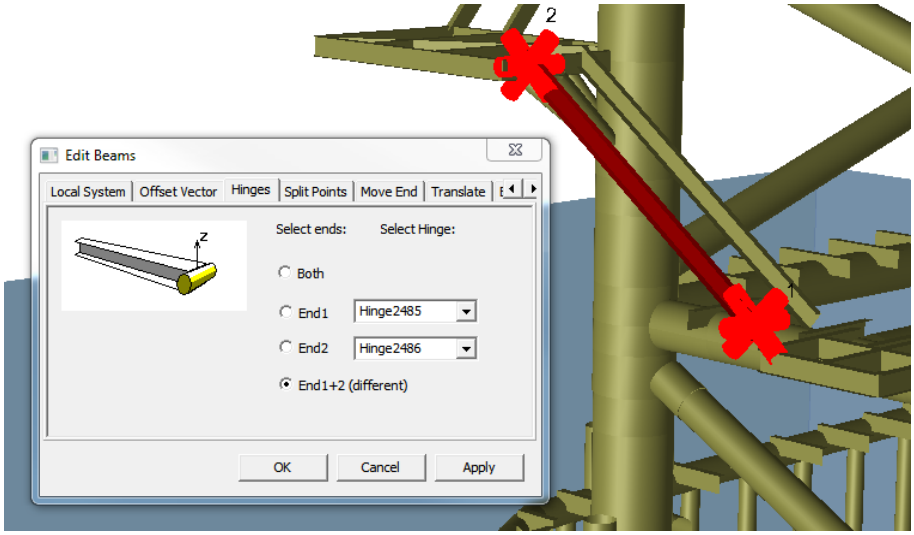


Figure 45: Hinges tab activates display of beam hinges

Go to the Properties→Hinges folder and select Edit Hinge on the Hinge2485. Change ‘rx’ to “Not hinged”. Hinge2486 does not have to be changed.

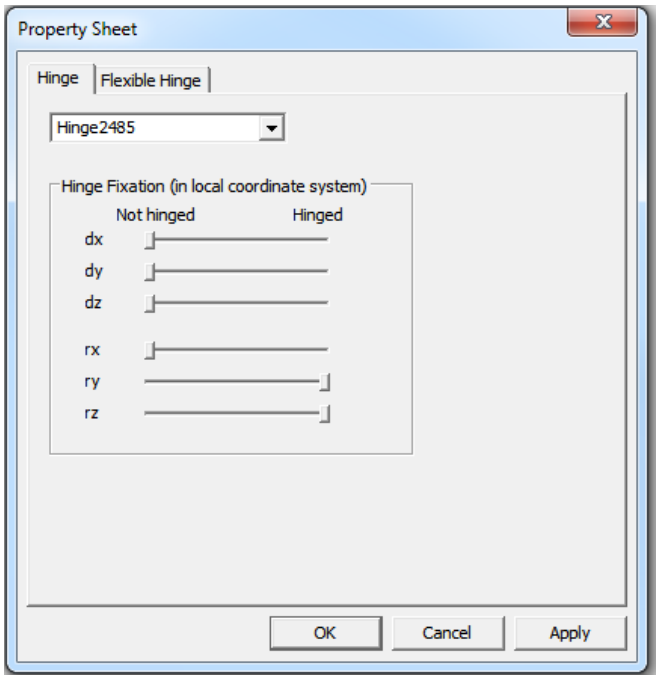


Figure 46: Change rotation around local X-axis to "Not hinged"

4.2.6 Element connected to same node twice

Linear Structural Analysis fails with the error message

```
*** ERROR *** Element    9256 is connected twice to the same node    2737
```

What to do:

Performing a ‘Locate FE…’ in ‘Mesh - All’ view shows the beam element causing the problem. Switch to Structure view and verify that a spring without a corresponding joint is created. This results in an illegal ConnectionPoint definition.

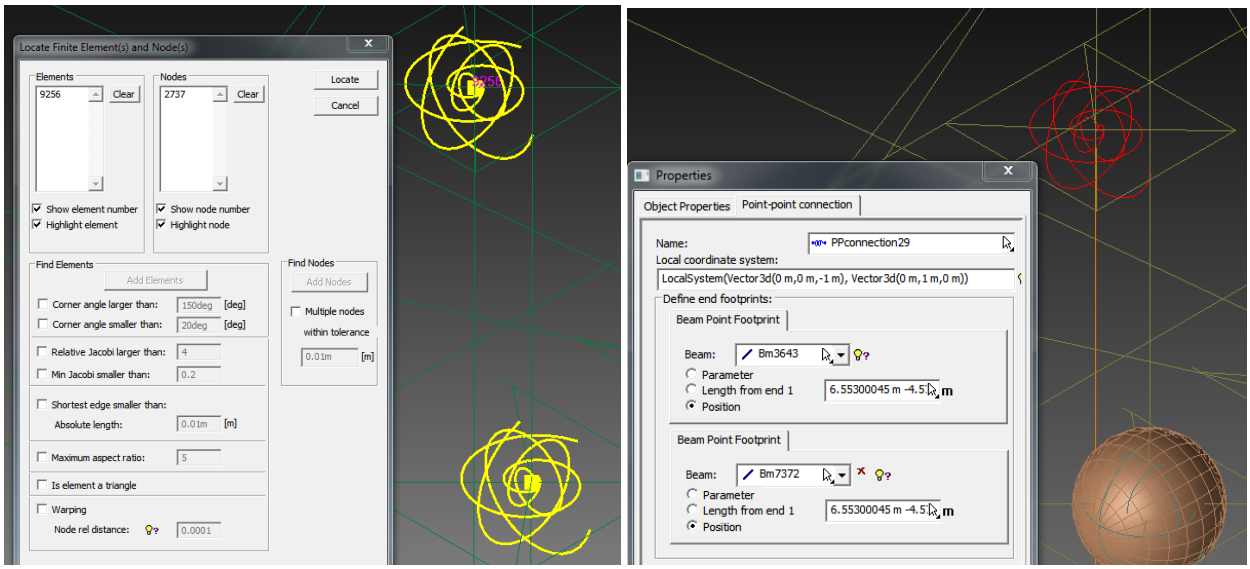


Figure 47: ‘Mesh – All’ view to localize element and ‘Modelling – Structure’ view to show spring

In this particular case the reason for the problem was an error in SACS input. The Z-coordinate of the JOINT that connects the four guide elements was inaccurate. When correcting the coordinate value in the SACS input the spring was correctly defined with a corresponding joint.

4.2.7 Pure DEAD load cases when structure is dry will make Sestra fail

If the SACS input file contains pure DEAD load cases and the structure is dry the Linear Structural Analysis fails with error messages of type

```
*** INPUT ERROR ***
      Loadcase(s) defined in LCOM command with identification no. =     6
      do not exist on Input Interface or Load Files.
      Load combinations in PRESEL may be wrong.
      The highest loadcase number found is =    12
```

The DEAD card of SACS includes dead loads and buoyancy in the load case. This card may also override the water depth of the model. An example may be to set the water level to zero in a pure DEAD load case to calculate dry weight of the structure. Such a pure DEAD load case produces two CalmSea states in GeniE with Buoyancy Only and Weight option, respectively. When the structure is dry, Wajac calculates no "wave load" forces for these states. The reason for Sestra failing is that the LCOM card of the sacstmpS1.FEM file refer to non-existing load cards.

What to do:

The way to solve the problem is to add a water level that is slightly above the lowest part of the structure and assign this depth to the two Calm sea states.

4.3 Pile-soil analysis fails

Several components are involved in a pile-soil analysis.

If Gensod fails the Gensod.LIS and Gensod.MLG files should be inspected. Look for error messages. Read the Gensod User Manual to get additional information. In particular see chapter 4 - Error Messages.

If Sestra-Reduction fails the Sestra_RED.LIS and Sestra_RED.MNT files should be inspected. Look for error messages. Read the Sestra User Manual to get additional information. In particular see chapter 4.4 Warnings and Error Messages.

If Splice fails the Splice.LIS and Splice.MLG files should be inspected. Look for error messages. Read the "Sesam-Splice/Interaction, Structure/Pile/Soil interaction analysis" User's Manual to get additional information. In particular see Appendix E - Error Messages.

If Sestra-Retracking fails the Sestra_RET.LIS and Sestra_RET.MNT files should be inspected. Look for error messages. Read the Sestra User Manual to get additional information. In particular see chapter 4.4 Warnings and Error Messages.

Note that the error handling has been improved from version 7.1 of GeniE/Gensod/Splice. Note the following:

Error messages compared to earlier versions of Splice have changed. Splice is now more restrictive and major problems that earlier was reported as warnings will now be treated as errors, as the results with warnings may be inaccurate. The error messages have been updated with additional info about the cause and how to correct the input. In most cases when error appears the user should (1) check that correct units are used in the different input files, and (2) try to increase the number of pile elements to reduce the soil layer thickness. In general an element length of 1 meter is recommended.

For this reason the description/error messages in some of the following chapters may be inaccurate.

4.3.1 Missing vertical fixations (Sestra-Reduction)

Sestra, Reduction may fail with the error message

```
      Error in factorisation of Stiffness matrix.
      Matrix is not positive definite.
      The error may be related to External Node no:     138
      Degree of freedom no:   3
      Coordinates of the node: -8.940E+00  8.940E+00 -4.013E+01
      Connected to External Element no:     144
      Connected to External Element no:     156
      Connected to External Element no:     161
      Connected to External Element no:     165
      Connected to External Element no:     185
      Connected to External Element no:     363
```

This is an indication of not well conditioned boundary conditions which can have several reasons. In this case the problem is missing vertical fixations at the top of piles. The model must be "washed" in several steps.

What to do:

Perform a 'Locate FE…' in 'Mesh - All' view. In this case the problem area is at a joint of the leg. This indicates that a vertical fixation at the top of the leg is missing.

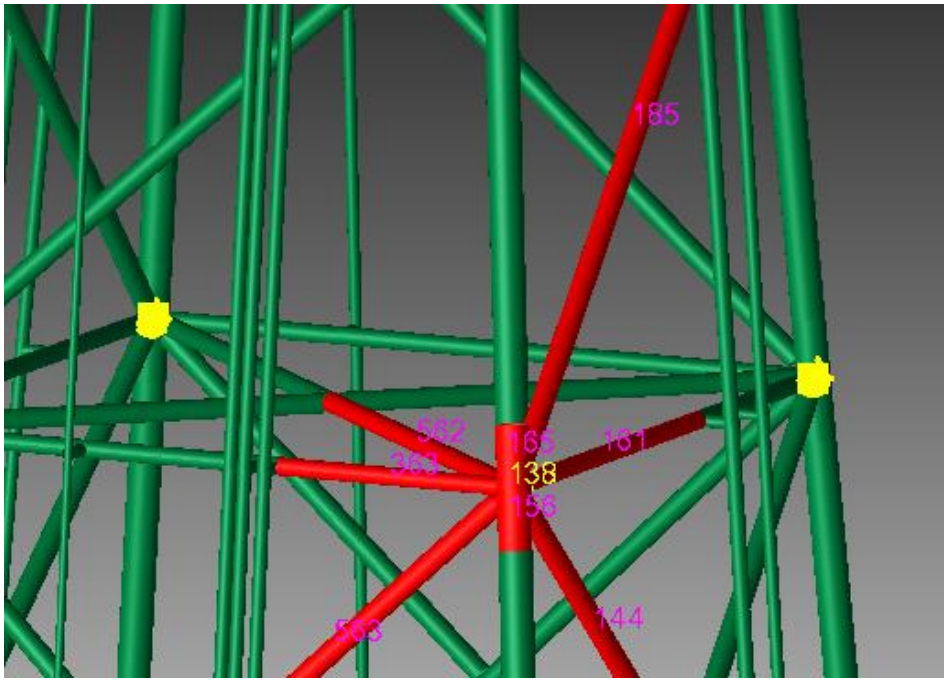


Figure 48: The problem area indicates that a vertical fixation at the top of the leg is missing

The vertical fixation can be achieved in several ways. One method is to detect the joint at the top of the leg and merge its connection points into one as described in 4.2.1. Additionally delete the spring at the same location.

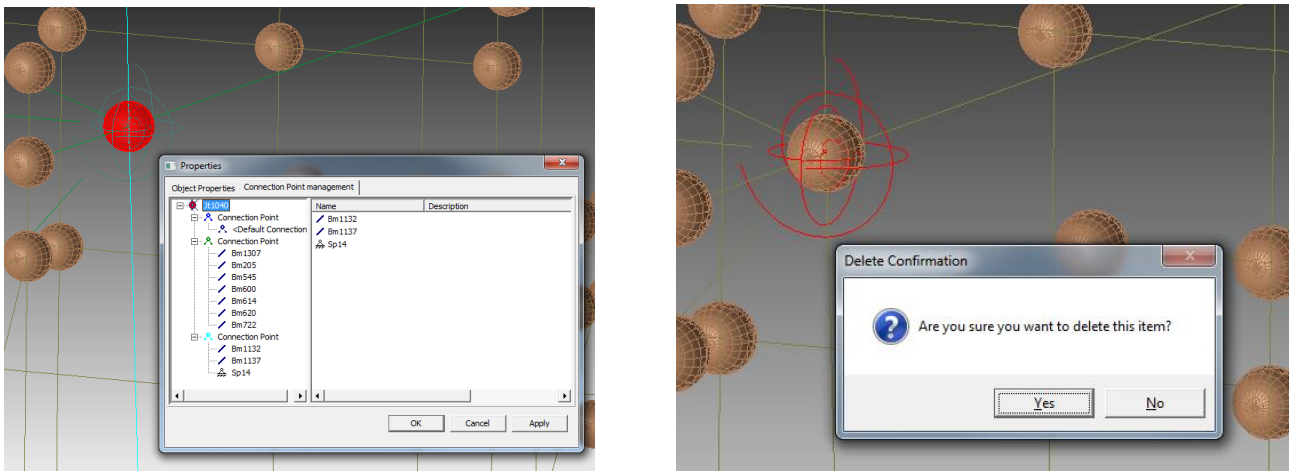


Figure 49: Make vertical fixation by merging connection points and deleting corresponding spring

It may be necessary to repeat these last two operations several times to remove all similar types of Sestra errors, i.e. once for each leg.

4.3.2 Bad connectivity between pile and leg, case 1 (Splice)

Splice may fail with the following error message

```
        ** WARNING ** NODAL STIFFNESS FOR SUPERSTRUCTURE NOT FOUND
                       WHEN READING FROM MATRIX INTERFACE FILE.
                       FILENAME: sacstmpM21.SIF
                       NODE I:       28
                       NODE J:       58
```

Further up in the same file the following information is found.

```
NUMBER OF SUPERSTRUCTURE SUPPORT JOINTS =     8
 SUP.STRUCT NODE   PILE HEAD NO.
          58             1
          28             2
.........
```

This indicates bad or missing connectivity between pile and leg due to that a part of the leg extends below the pile head. In this particular case it is a thin slice, only.

What to do:

Perform a graphical check and delete part of the leg that extends below the pile head (pile skirt).

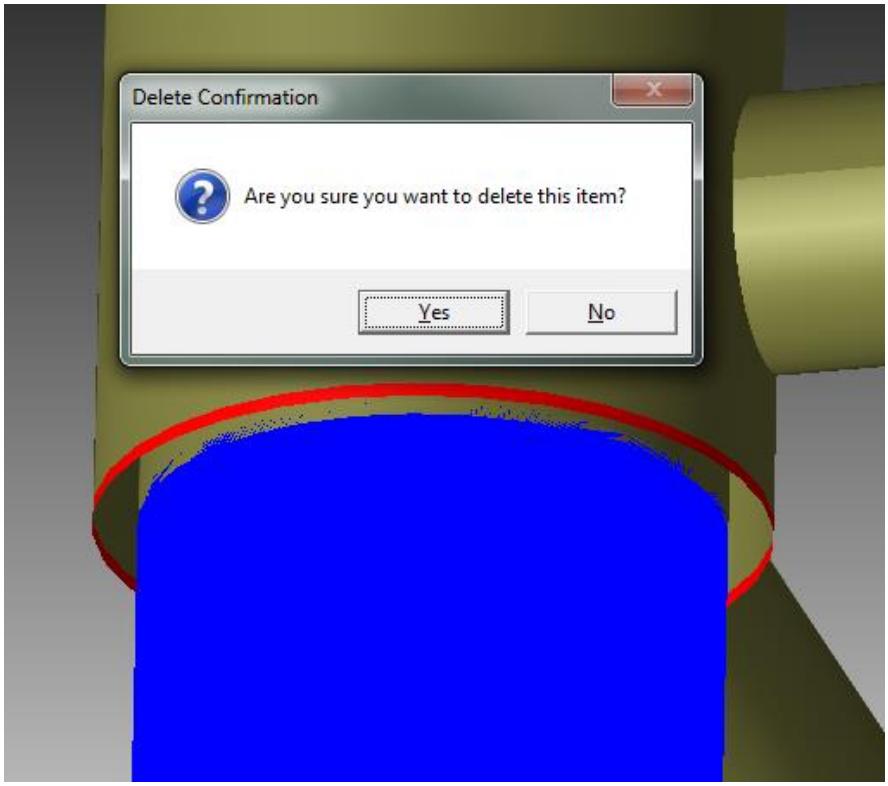


Figure 50: Delete parts of the leg extending below pile head (the pile is blue)

When the "pile skirts" are deleted a new error message from Splice occurs in this particular example.

```
       *** ERROR *** This pile is not connected to any node in the structure.

                     ERROR/WARNING STATUS IS:       -1
                     PILE NUMBER:       1
                     ERROR/WARNING IN ROUTINE PILEIN_PRFR
```

This indicates missing connectivity between pile and leg due to illegal geometry at the pile head.

What to do:

Perform a graphical check and delete illegal geometry at the pile head. In this case a narrow beam overlaps the lower, inner part of the leg, i.e. the pile extension of the leg.

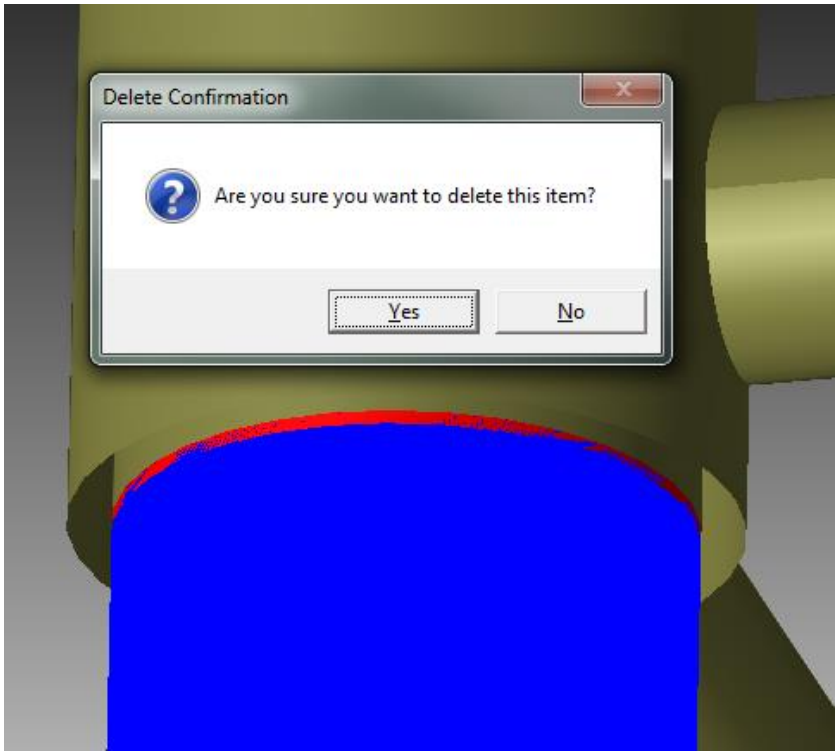


Figure 51: Delete illegal geometry at pile head (the pile is blue)

4.3.3 Bad connectivity between pile and leg, case2 (Splice)

Splice may fail with the error message as described in the previous chapter, but the actions described do not help. The most likely explanation is that the "Wishbone name mask" value in the import SACS dialog is wrong.

What to do:

The default "Wishbone name mask" is "W.B". Inspect the CHK file and look for messages of type:

…

```
Warning: member with group label WB  is a wishbone candidate but does not comply with
wishbone name mask W.B. Member is rejected.

MEMBER1527 526  WB          100111

Warning: member with group label WB1 is a wishbone candidate but does not comply with
wishbone name mask W.B. Member is rejected.

MEMBER1310 309  WB1         100111
```

…

They indicate that the name mask should be changed to "WB*". The correct name mask may also be decided by inspecting the GRUP cards of the SACS input file.

4.3.4 Bad connectivity between pile and leg, case3 (Splice)

Sestra, Retracking may fail with the following message

```
** WARNING ** RESULTING DISPLACEMENTS FROM 'Splice' NOT FOUND,
                WHEN READING FROM MATRIX INTERFACE FILE.
                ERROR/WARNING STATUS RETURNED FROM ROUTINE GRES73,
                CALLED FROM PILE91. STATUS FLAG =       0
```

```
  *** Error occurred during reduction or retracking.
```

In this case Splice has only issued a warning message. When the Splice.MLG file is inspected it may show a message similar to

```
        ** WARNING ** NO. OF NODES ON MATRIX INTERFACE FILE IS NOT EQUAL TO NO. OF
                      PILE NODES PLUS CONNECTION NODES IN STRUCT. FROM 'T-FILE'
                      FILENAME: sacstmpM21.SIF
                      NO. OF NODES ON 'M-FILE':     264
                      PILE+CONN. NODES ON 'T-FILE':       0
                      ERROR/WARNING STATUS RETURNED FROM ROUTINE              ,
                      CALLED FROM ZERJAC_PRFR  . STATUS FLAG =        0
```

This may indicate bad or missing connectivity between pile and leg. In this particular case it is caused by structural beams overlapping parts of the piles.

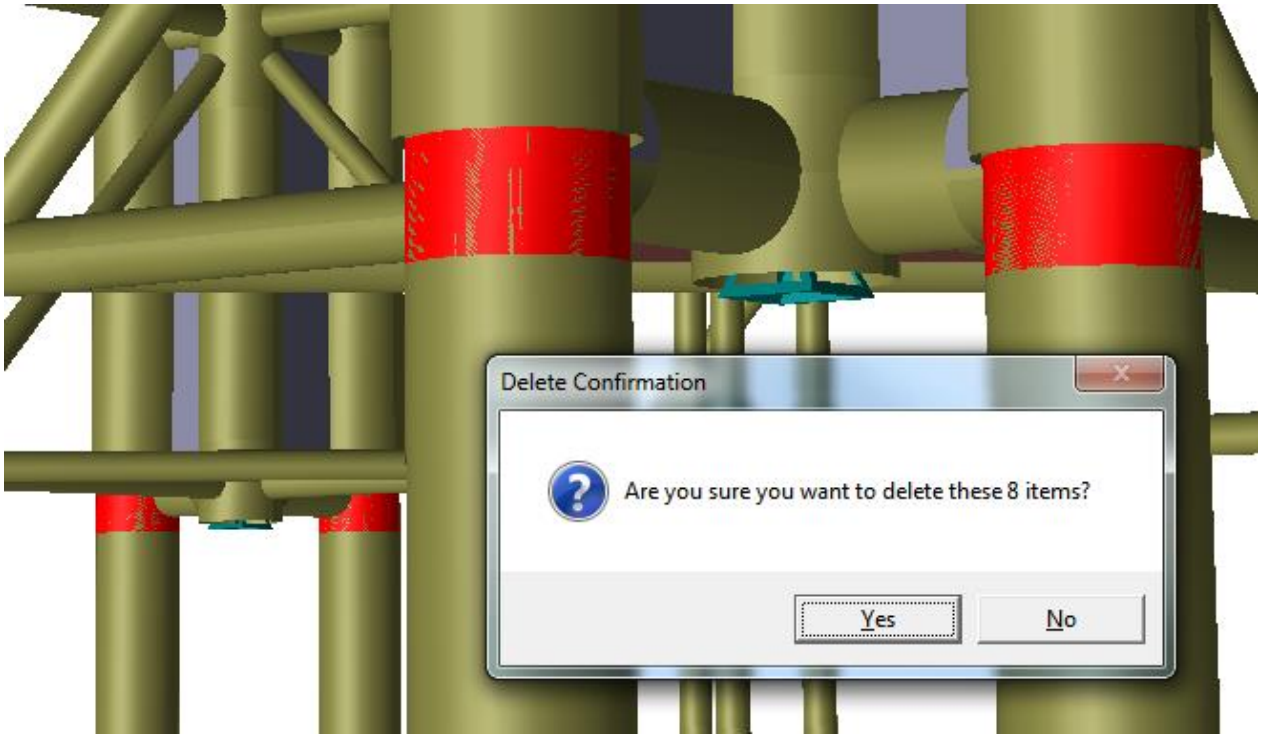


Figure 52: Overlapping parts of the pile make Splice fail. Deleting these parts solves problem.

What to do:

Deleting the overlapping parts below mudline as shown in Figure 52 solves the problem.

4.3.5 Inner piles not properly fixed against torsion, solution 1 (Splice)

Splice may fail with the following error message

```
....
 ***** ERROR MESSAGE :      1210        4        9  -7.431E+00
 ***** ERROR MESSAGE :      1210        4        9  -7.431E+00
```

```
***** ERROR MESSAGE :      1420       48        8   -1.717E+01
EXECUTION OF THIS RUN TERMINATED :  28-MAY-2013   11:50:53 HOURS
LAST PILE SOLUTION ON FILE :    13
TIME USED =                10 SECONDS
ERROR EXIT FROM Splice
```

The **1210** error indicates 'Poor Torsional Modelling'. It is seen that extreme combinations of slender piles and rigid soils may result in this kind of numerical problem. There are several ways of solving this problem. In this chapter a solution based on the proposal in the "Sesam – Splice, User's Manual" stating that "the element height must be reduced" is described.

What to do:

Open the Soil tab of the "Edit Location" dialog by "right clicking" on the Location1 object in the browser and select "Edit Location". Inspect the soil model and insert a proper number of sublayers within each layer. Alternatively, check the "Refine soil layers" (Soil tab) option in the "Import SACS" dialog to control the layer thickness during import. Read more about how to use this option in chapter ~~9.6.4~~ 10.6.3 in Volume 3 of the GeniE UM.

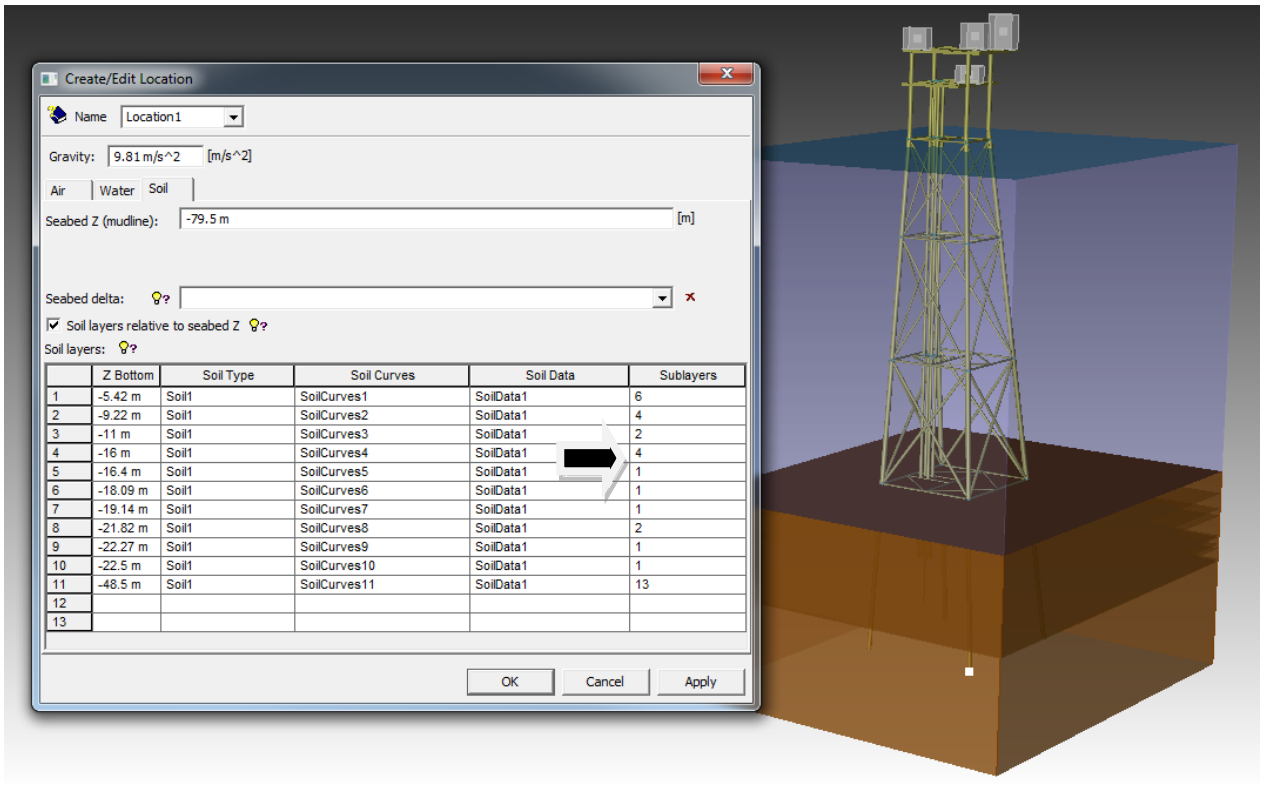


Figure 53: Insert a proper number of sublayers within each soil layer

4.3.6 Inner piles not properly fixed against torsion, solution 2 (Splice)

Splice may fail with the following error message

```
....
***** ERROR MESSAGE :      1210        4        9   -7.431E+00
***** ERROR MESSAGE :      1210        4        9   -7.431E+00
```

```
***** ERROR MESSAGE :      1420       48        8  -1.717E+01
EXECUTION OF THIS RUN TERMINATED :  28-MAY-2013   11:50:53 HOURS
LAST PILE SOLUTION ON FILE :   13
TIME USED =                10 SECONDS
ERROR EXIT FROM Splice
```

The **1210** error indicates 'Poor Torsional Modelling'. It is seen that extreme combinations of slender piles and rigid soils may result in this kind of numerical problem. There are several ways of solving this problem. In this chapter a solution based on modifying the Pile Characteristics is given.

What to do:

Open the "Edit Pile Characteristics" dialog by "right clicking" on the PileType1 object in the browser and select "Edit Pile Characteristics". Check the "infinitely long beneath Z" option and give a proper Z-value.

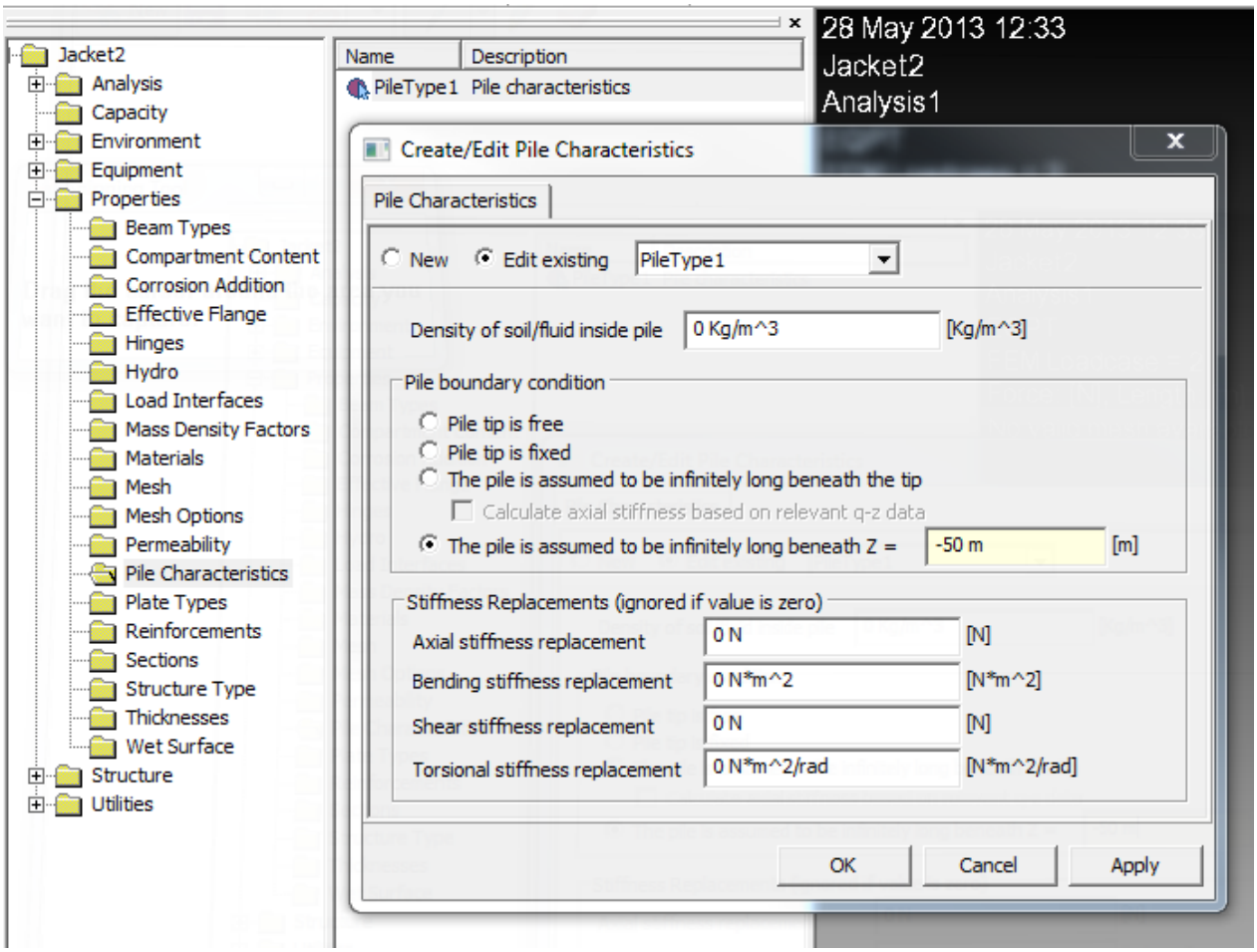


Figure 54: Select the "infinitely long beneath Z" option

4.3.7 Inner piles not properly fixed against torsion, solution 3 (Splice)

Splice may fail with the following error message

```
....
***** ERROR MESSAGE :      1210        4        9  -7.431E+00
***** ERROR MESSAGE :      1210        4        9  -7.431E+00
```

```
***** ERROR MESSAGE :     1420       48        8   -1.717E+01
EXECUTION OF THIS RUN TERMINATED :  28-MAY-2013   11:50:53 HOURS
LAST PILE SOLUTION ON FILE :   13
TIME USED =                10 SECONDS
ERROR EXIT FROM Splice
```

The **1210** error indicates 'Poor Torsional Modelling'. It is seen that extreme combinations of slender piles and rigid soils may result in this kind of numerical problem. There are several ways of solving this problem. In this chapter a solution based on inserting joints and manually correct connections is described.

What to do:

In this case the solution is to add a joint on top of the leg and make sure that the correct beams are connected. After having inserted the joint, "right click" it and select "Properties". Then you have to move the connected beams to the correct connection point. This may be necessary to do for several legs.

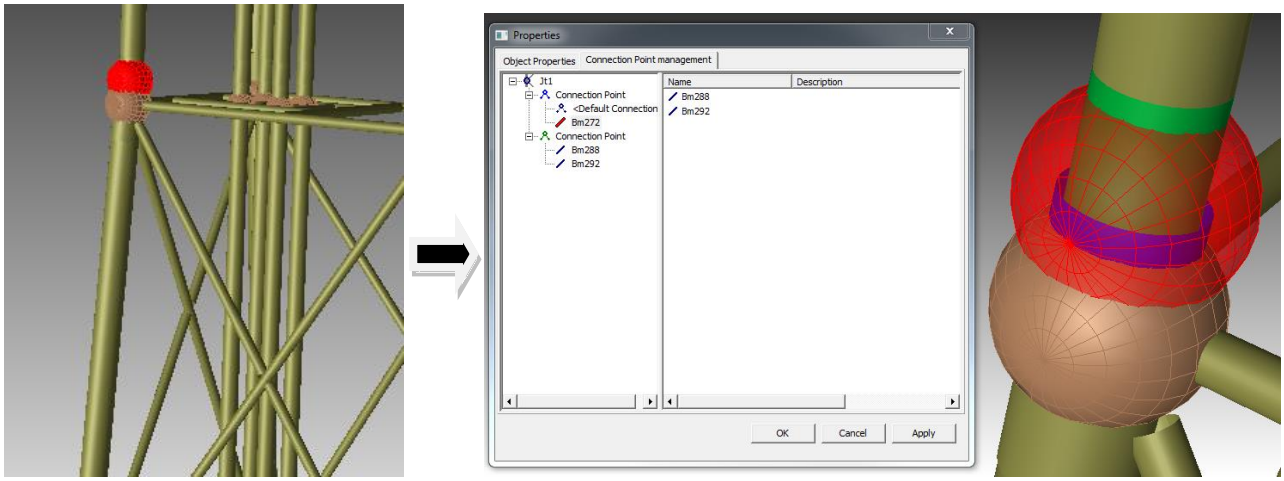


Figure 55: Add a joint on top of the leg and manually modify the connectivity and ensure correct connectivity by using the Connection Point management.

4.3.8 Splice fails with error message 1050

If Splice fails the Splice.MLG file should be inspected. If the error message is

```
***** ERROR MESSAGE :     1050       2        2    3.874E-02
```

The **1050** error indicates that there is a numerical problem in the pile system. Numerical problems may arise from too many elements, too slender/flexible pile elements compared to G (shear module) for soil, too stiff soil, too weak piles or other reasons.

What to do:

Read the "Sesam – Splice, User's Manual". One way of avoiding this numerical is to define a Pile Characteristic where the pile is assumed to be infinitely long beneath a Z level 15 – 30 meters below mudline. Assign this property to the piles.

4.3.9 Splice fails with error message 1110

If Splice fails the Splice.MLG file should be inspected. If the error message is

```
***** ERROR MESSAGE :     1110       2       19   -1.083E+01
```

The **1110** error indicates 'Poor Axial Modelling'.

What to do:

Read the "Sesam – Splice, User's Manual". When this error message occurs, something is probably totally wrong with the input. However, it can be seen that even if this error occurs on the MLG file, the calculations can converge and the execution is successful.

4.3.10 Splice fails with error message 1210

If Splice failes the Splice.MLG file should be inspected. If the error message is

```
***** ERROR MESSAGE :     1210       2      21  -4.149E+00
```

The **1210** error indicates 'Poor Torsional Modelling'. It is seen that extreme combinations of slender piles and rigid soils may result in this kind of numerical problem.

What to do:

Read the "Sesam – Splice, User's Manual". In these cases the element height must be reduced by increasing number of sublayers. This can either be done from the Soil tab in the 'Import SACS' dialog or from the Soil tab of the 'Edit Location' dialog.

4.3.11 Splice fails with error message 1310/1320

If Splice fails the Splice.MLG file should be inspected. If the error message is

```
***** ERROR MESSAGE :     1310       2      11  -3.835E+00
 ***** ERROR MESSAGE :     1320       2      11  -3.835E+00
 ***** ERROR MESSAGE :     1010       2       3   4.371E+21
```

The **1310/1320** errors indicate 'Poor Lateral Modelling' while the **1010** error indicates 'Lateral matrix not invertible'. It is seen that this error may occur when the elements are too small. Too small elements may occur if the layers are thin. In Figure 56 is shown an example of soil layers and elements generated with the default "Minimum layer thickness" of 0.1 m that gives this error.

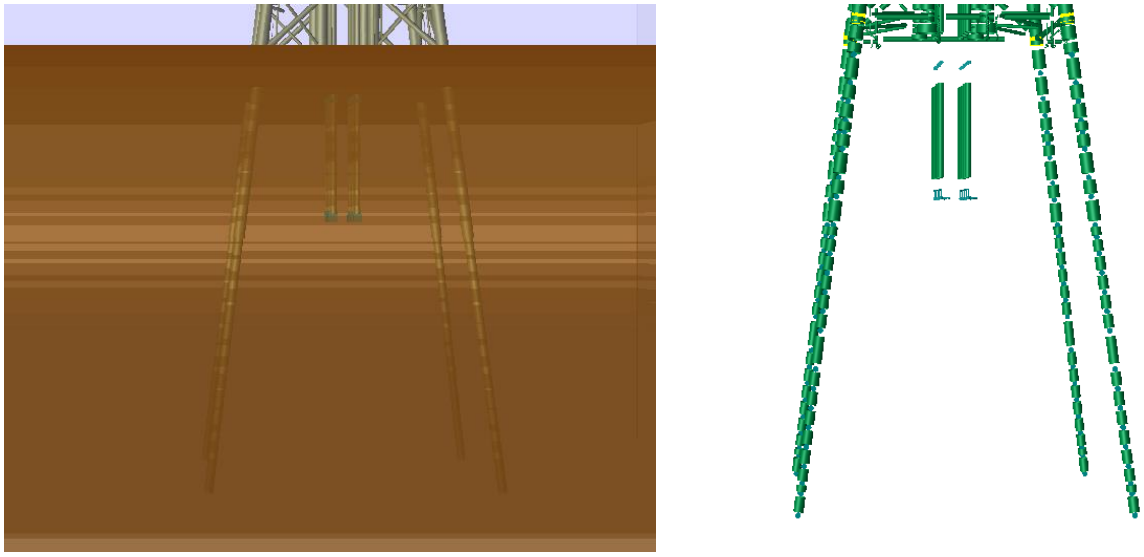


Figure 56: Layers and elements generated with default minimum layer thickness

What to do:

Read the “Sesam – Splice, User’s Manual” for more information. In this case it is recommended that the element height is increased by increasing the “Minimum layer thickness” value in the Soil tab of the “Import Sacs File” dialog. In Figure 57 is shown the soil layers and elements generated when increasing the value from 0.1 m to 4.0 m. In this case Splice executes successfully.

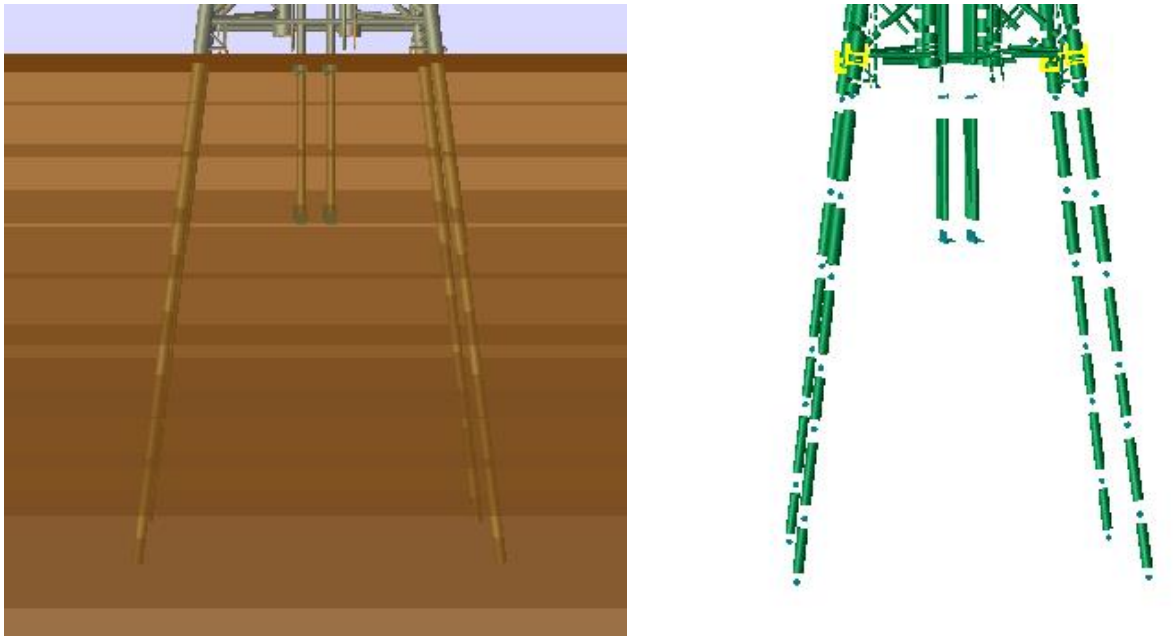


Figure 57: Layers and elements generated with modified minimum layer thickness

5. OTHER ISSUES

5.1 Marine growth density

The marine growth density may seem different in GeniE compared to the SACS input as shown below.

Sacs input file

```
CDM
CDM     10.0   0.65   0.008 1.6          1.05   0.008 1.2
CDM    180.0   0.65   0.008 1.6          1.05   0.008 1.2
WINSHL         9.5     17.75
MGROV
MGROV        0.000   15.81     2.5                      1.4
MGROV       15.81    21.81     3.0                      1.4
*
*
```

GeniE

MarineGrowthConstant1 Marine growth values, Thickness=0.025, Roughness height=0, Use for inertia force calculations=No, Density factor=1.36
MarineGrowthConstant2 Marine growth values, Thickness=0.03, Roughness height=0, Use for inertia force calculations=No, Density factor=1.36

The reason is that in the SACS input file, the marine growth density is specified. In GeniE, a marine growth density factor is used, the factor giving the density of marine growth relative to sea water density.

Example:

Marine growth density = 1.400 tonnes/m^3

Sea water density = 1.032 tonnes/m^3.

Marine growth density factor = 1.400 / 1.032 = 1.36

5.2 Weight of marine growth

A SACS loading condition may contain wind data, only.

If GeniE 7.0 and earlier version is used, the following applies:

When imported to GeniE this results in a general load condition with Wind Field activated and a “CalmSea” seastate with the wind profile assigned. Note that even if buoyancy calculations are turned off, weight of marine growth is calculated. The load will be based on a net density equal to “density of marine growth” minus “density of water”.

If GeniE 7.1 or newer versions is used the same two load conditions are created, one general load condition with Wind Field activated and a “CalmSea” seastate with the wind profile assigned. In this CalmSea seastate the option of buoyancy is turned off and there is no marine growth weight included if the option of “Include weight of marine growth” is not checked.

5.3 Limitation in load combinations

Some limitations in load combinations are described below.

5.3.1 Combinations including a pure wind condition may get wrong load factors

As mentioned above a SACS loading condition may contain wind loads, only. If such conditions are defined before wave load conditions in the SACS input file, load combination that contain these wind and wave conditions may get wrong load factors. *This problem exists for GeniE-version 6.7 and older.*

Workaround

Edit the SACS input file and move the wind conditions after the wave conditions.

5.3.2 Combinations including a pure wind and a wave condition gives wrong weight and buoyancy of marine growth

If a load combination consists of a pure wind condition and a wave condition the combinations may look as follows:
`Analysis1.WLC(1,1) x 1.3 + Analysis1.WLC(3.1) x 1.1`
where Analysis1.WLC(1,1) is the wave condition and Analysis1.WLC(3.1) is the pure wind (calmsea) condition.

Weight and buoyancy (net load contribution) of marine growth is always calculated with any one of options BuOn, BuOff and BuOnly selected. When Analysis1(1,1) and Analysis1(3,1) are combined, the effect of marine growth is calculated for both load cases.

Note that this only will give contribution if the marine growth density adjustment factor (compared with density of water) is different from 1.0.

This won't happen in GeniE 7.1 and newer versions if the option of "Include with of marine growth" is not checked.

5.3.3 Combinations including an AL wave condition are created one for each step

A wave condition (WAVE card) in the SACS input file with the "Critical Position"-value equal to AL will result in one wave load case for each wave step in GeniE. A load combination referring to this wave condition cannot contain more than one wave load case in GeniE. Thus, one load combination must be created for each wave step.

The naming convention and content of the combinations are as follows

If the WAVE card (of loading condition W001) defines 4 wave steps, the wave loads in GeniE will typically look like `Analysis1.WLC(1,1),…, Analysis1.WLC(1,4`. In addition a dummy wave load case named W001 will be generated. If a load combination (LCOMB card) C001 refers to W001, the load combinations in GeniE will look like `C001_1,…,C001_4`. For example will C001_1 refer to Analysis1.WLC(1,1) and W001, while C001_3 will refer to Analysis1.WLC(1,3) and W001.

5.4 The non-structural KEEP and DELETE sets

The DUMMY/KEEP/DELETE cards describe connected non-structural elements (dummy structure). Each group of these cards are converted to a pair of sets in GeniE; KD_<name>_KEEP and KD_<name>_DELETE, where <name> equals the unique name of the dummy structure defined by the DUMMY card.

The KEEP set contains joints where the dummy structure connects to the elastic structure, while the DELETE set contains beams and plates of the dummy structure. These two sets enable easy identification of secondary parts.

There are several ways to proceed in GeniE. These are described below.

1. Connect structure by use of short members, typically for appurtenances. Note that normally this should not be relevant, because these sets describe connected non-structural parts. However, there are several examples of unconnected non-structural parts described by these sets as well.
2. Point-point management by use of spring characteristics between two points.

3. Define members of the DELETE sets as non-structural. Define linear dependencies, i.e. master-slave techniques. This option is the most efficient when there are many non-structural members, e.g. like in boat landings, as several members can be connected as a volume to the same master node. The connections will be done using 6 degrees of freedom (user may change to fewer DOFs) and the user may modify the stiffness characteristics within the limits of singularity.
4. Convert non-structural to structural beams, i.e. delete the non-structural Beam type property. Reduce the E-modulus of the material for the dummy structure, e.g. by 100.

5.5 The RES_GRUP set

GRUP card specifies in column 47 (value "9") if the members shall not be included in stress calculations. A set named RES_GRUP is created during the import. This set contains all members that contribute in result view or code check, i.e. the complementary set of members.

5.6 The RES_GRUP_MOD set

This set equals the RES_GRUP set excluding all members of the DELETE sets described above.

5.7 The CDWN_TUB, CDWN_NTB, CDWN_SHL and CDWN_ALL sets

The Member Loading Option on the WIND card of the SACS input file describes how wind load is applied on members. The following description is valid for GeniE v7.1 and older.

- If the option has no value (blank), tubular and non-tubular members above water line shall have different air drag coefficients. The CDWN_TUB and CDWN_NTB sets contain the respective members. Members of the CDWN_TUB set are assigned an air drag coefficient equal to 0.5 and members of the CDWN_NTB set are assigned an air drag coefficient equal to 1.5.
- If the option is 'I', no members shall have wind load. In this case all members above water line are assigned an air drag coefficient equal to 0.0. The CDWN_ALL set contains these members.
- If the option is 'W', SACS says that air drag coefficients of tubular members above water line are to be the same as used for the Steady State Current. In this case GeniE utilizes the default air drag coefficients of Wajac. These coefficients are a function Reynold numbers (according to Wajac manual, section 2.4.9) and are applied to all members above water line. A similar message as described below is given for GeniE v7.1 and older.
- If the option is 'W' or blank and the SACS input file contains a WINSHL card, describing from 1 to 4 wind shielding zones, a CDWN_SHL set is made. The set contains all members within these zones. These members are assigned an air drag coefficient equal to 0.0.
- Note that in SACS the WIND card is part of a loading condition (LOADCN) and wind forces and air drag coefficients are tailored accordingly. However, in GeniE the air drag property is common to all load cases. Therefore, during import the Member Loading Option value of the WIND cards are counted and the option with the highest count is used.

For GeniE v7.2 and newer the description above is valid, except for the following.

- If the option is 'I', the wind profile is not assigned to its wave load case. In this way the 'I' option is given a local behaviour, i.e. it behaves as in SACS. Note that if there are no CURR or WAVE lines in the loading condition (LOADCN), no wave load case is created.
- Due to this enhanced handling the CDWN_ALL set will no longer be created.
- If the SACS input file contains both 'W' and 'no value' for Member Loading Option, the option with the highest counting will decide - as before. If the count is equal, the 'no value' behaviour is selected.
- If the 'W' option is the dominating the following message is given in Message tab/LOG file

```
Warning: Member Loading Option value W on the WIND cards is the
dominating and wind forces will be calculated using default air
drag coefficients of Wajac if no action is taken by the user. See
chapter 5.7 of the Guidelines document for more info.
```

- **In general, if the 'W' option is the dominating the user is advised to define air drag coefficients the same as Cd values for wave/current force calculations.**

5.8 The PLATE_TYPE sets

When *Defined* is selected as the "Import plate elements as" option a set is created for each PLATE TYPE defined in the SACS input file by the PGRUP cards. In the table below the set name related to each plate type is given, along with a comment.

Plate type	Set name	Comment
I or <blank>	PLATE_TYPE_I	Isotropic plate. Converted to a Shell element with a Linear Isotropic material.
M	PLATE_TYPE_M	Membrane plate. Converted to a Membrane element with a Linear Isotropic material.
S	PLATE_TYPE_S	Shear stiffness only. Converted to a Membrane element with a Linear Isotropic material. User should manually create a Shear Isotropic material and assign to all plates in set in GeniE.
X	PLATE_TYPE_X	Corrugated plate in X-direction. Converted to a Shell element with a Linear Isotropic material. User should manually create an Orthotropic material and assign to all plates in set in GeniE.
Y	PLATE_TYPE_Y	Corrugated plate in Y-direction. Converted to a Shell element with a Linear Isotropic material. User should manually create an Orthotropic material and assign to all plates in set in GeniE.

Table 1: Relation between Plate type and Set name in GeniE

Note that similar information to what is given in Table 1 is found in the CHK file.

5.9 How to identify the wave load condition that correspond to a WAVE card

If illegal values are given in the WAVE card messages as shown below may appear in the Message tab of GeniE and in the LOG file:

```
Warning: Zero value for step size/no.of steps value in LOADCN     75
```

In the CHK file a relation table between SACS LOADCN names and GeniE FEM load cases is found. It may show the following information.

```
LOADCN  75                  60
```

However, when a SACS loading condition contains WAVE cards they will results in Wave load conditions in GeniE. A Wave load condition name is not related to the LOADCN name. Neither there is a relation table in the CHK file between SACS LOADCN names and GeniE Wave load conditions.

To identify the Wave load condition that originates from the card shown below do the following.

```
WAVE
WAVE1.02STRN  2.8          6.92         315.0        D   0.00  0.00   0MM10 1 0
```

1. Open the “Edit Wave Load Run” dialog by RMB-clicking on the Wave Load Analysis step and select “Edit Wave Load Analysis”.

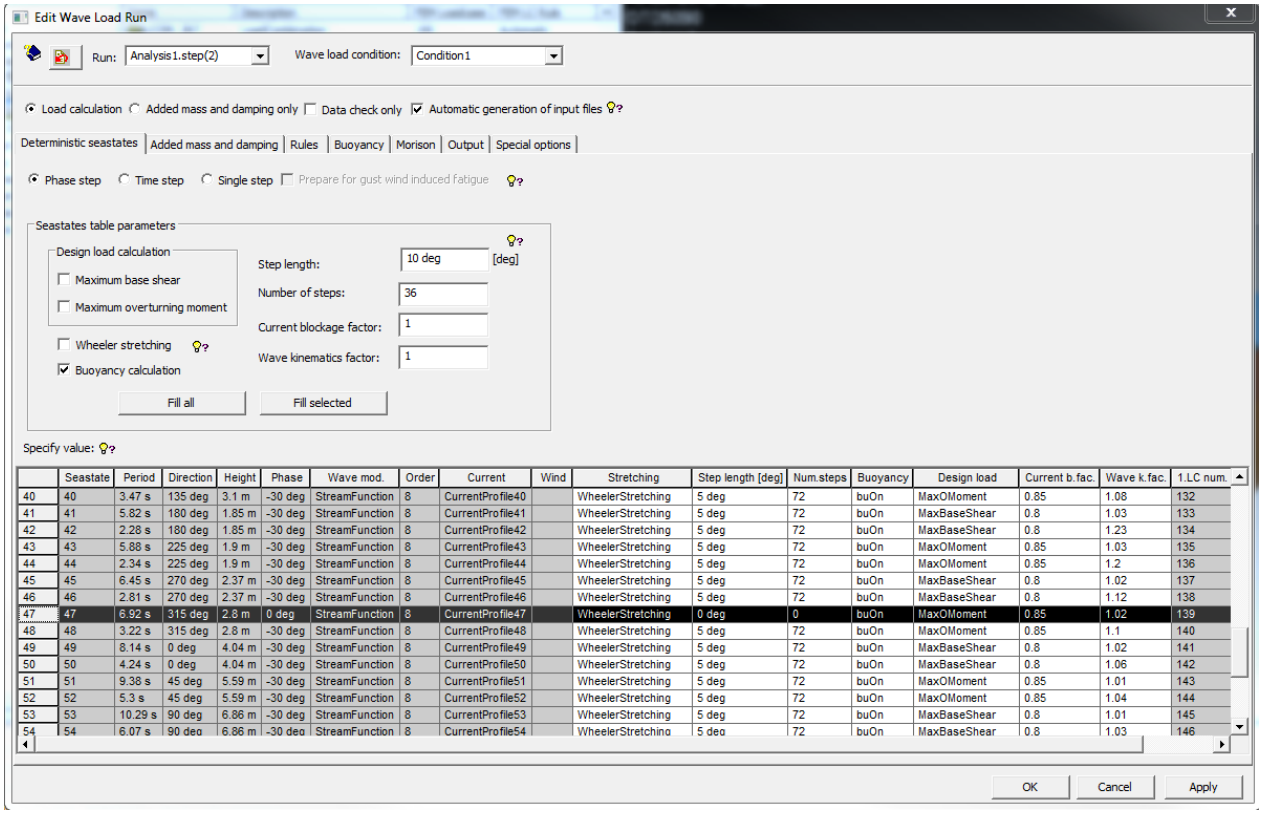


Figure 58: The Edit Wave Load Run dialog

2. Identify the Seastate with Period/Direction/Height values that corresponds to values in the WAVE card. The FEM loadcase name that corresponds to the wave card is found in the rightmost column in the dialog, “1.LC num.”.

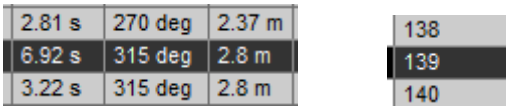


Figure 59: Identify the seastate by looking for correspondig values to the WAVE card. FEM loadcase name given in rightmost column.

5.10 Gravity load case with Buoyancy (GeniE v7.0 and older)

The following is valid for GeniE version 7.0 and older. When a load condition consist of only a DEAD card, i.e. a gravity load condition, the below warning will be shown after import.

```
Warning: LoadCase    1 includes gravity. Check that all parameters have been passed
since the gravity card may include details of buoyancy which cannot be consistently
converted.
```

In GeniE buoyancy will not be calculated in a gravity load case. If it is required that buoyancy is considered in a gravity load case, a load combination that includes buoyancy must be made. This is done by adding a CalmSea state and combine with gravity.

What to do:

Edit the Condition under Location, add a CalmSea for Wave model.

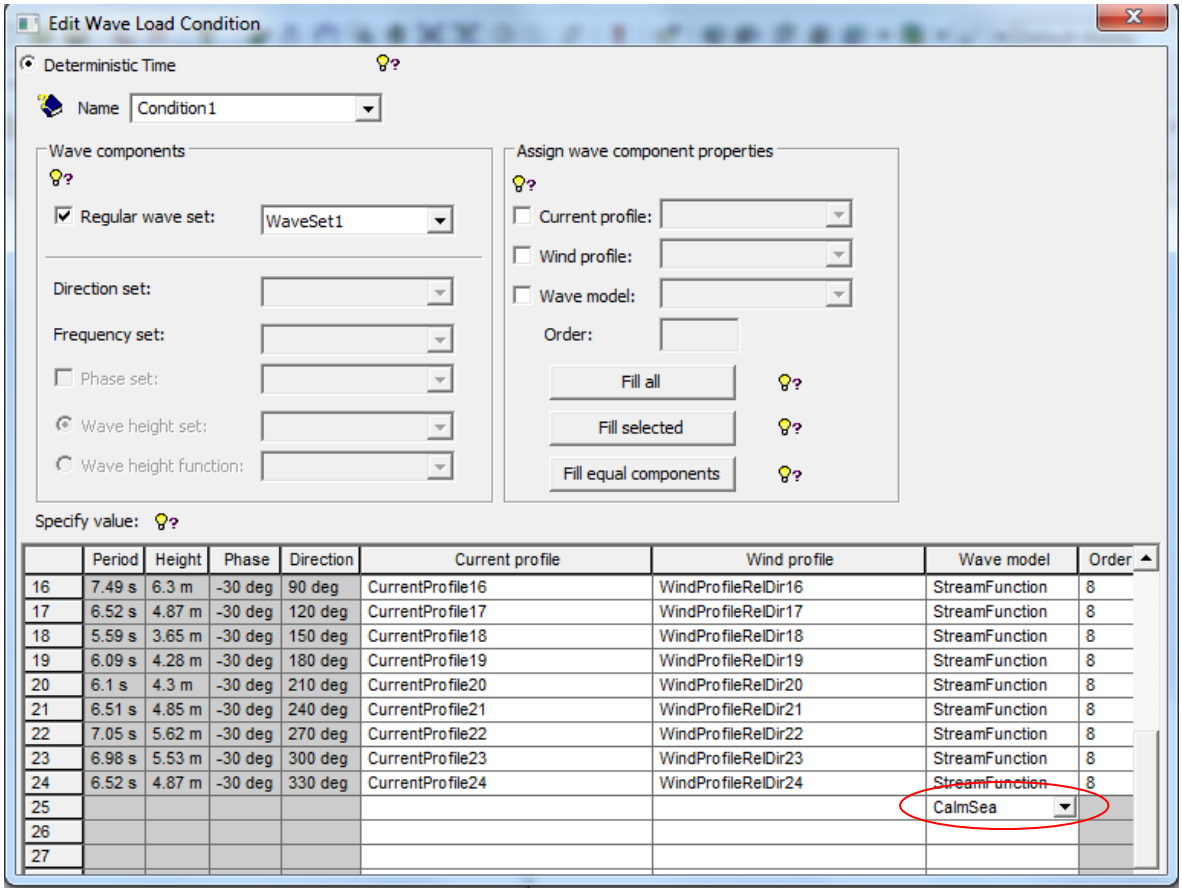


Figure 60: Add a Calm Sea state in the Edit Wave Load Condition dialog

The CalmSea will show in the seastate table of the “Edit Wave Load Run” dialog. Set buoyancy to ‘buOn’ for the CalmSea condition.

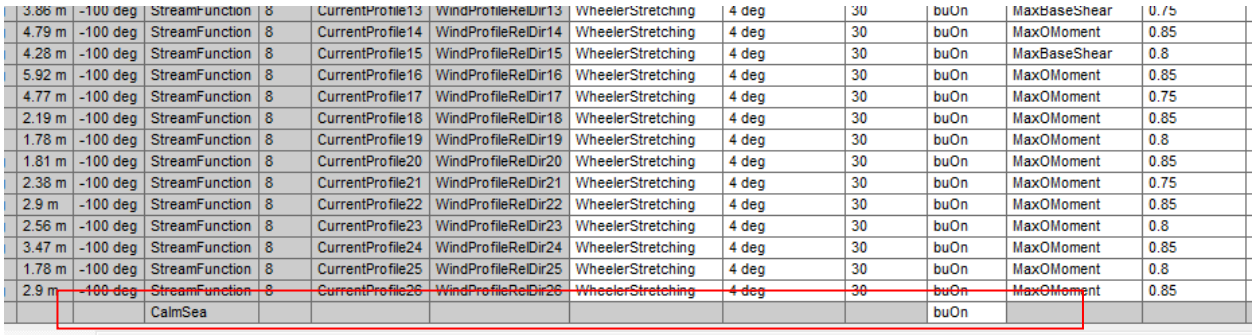


Figure 61: A CalmSea appears in the seastate table of the "Edit Wave Load Run" dialog.

Make a load combination of the gravity load case (LC1) and Calm sea load case (Analysis.WLC(25.1). Possibly substitute LC1 with LMG in relevant load combinations. In this case, to avoid contribution from buoyancy twice, switch to ‘buOff’ for the wave load conditions.

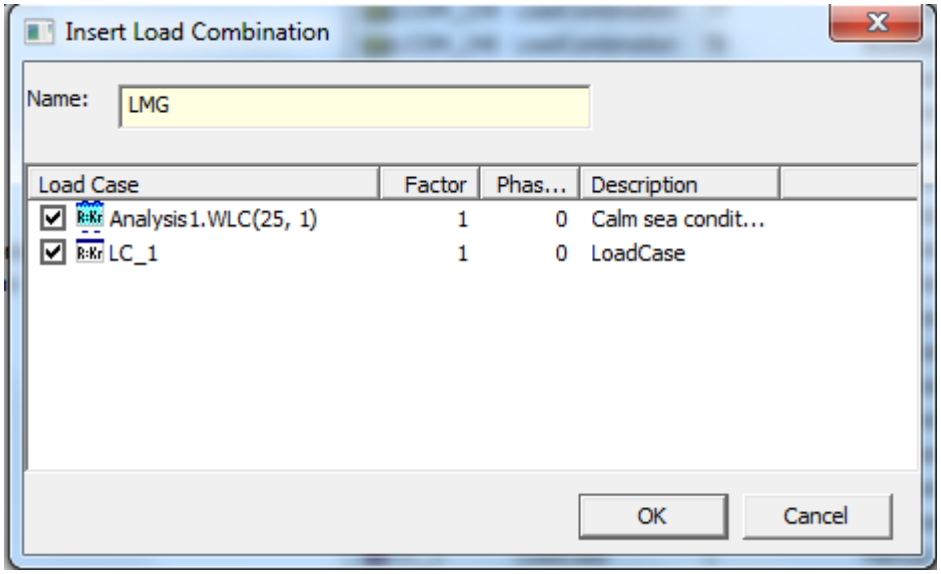


Figure 62: Make a new load combination of gravity load case and calm sea load case

5.11 Buoyancy load comparison

The buoyancy load calculation result could be found in Wajac.LIS. If the results are different compared to Sacs, the flooding and/or buoyancy area of beams need to be checked.

However, in some cases the difference may also come from the bottom elevation of the jacket. If some beams are in the mudline, as shown in Figure 63, the buoyancy of these will be not considered in GeniE by default. In Sacs they have been counted.

To activate calculation of buoyancy of such beams the “Include buoyancy of beams at mudline” check box of the Buoyancy tab on the “Wave Load Run” dialog must be selected as shown in Figure 64.

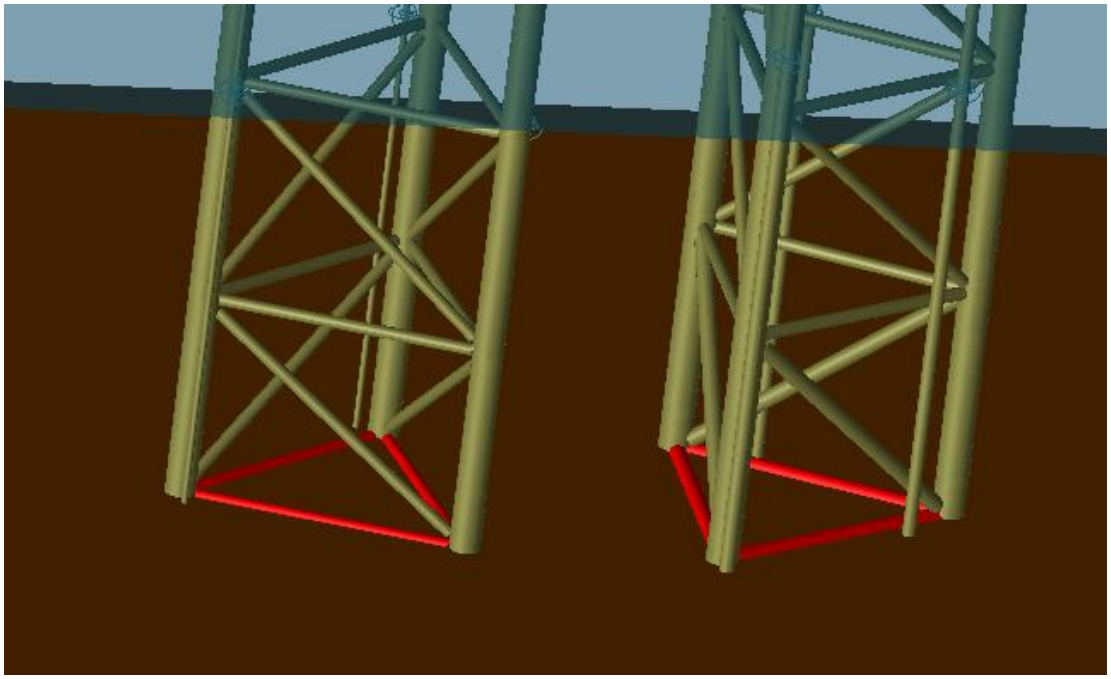


Figure 63: By default buoyancy of beams in the mudline are not considered in GeniE

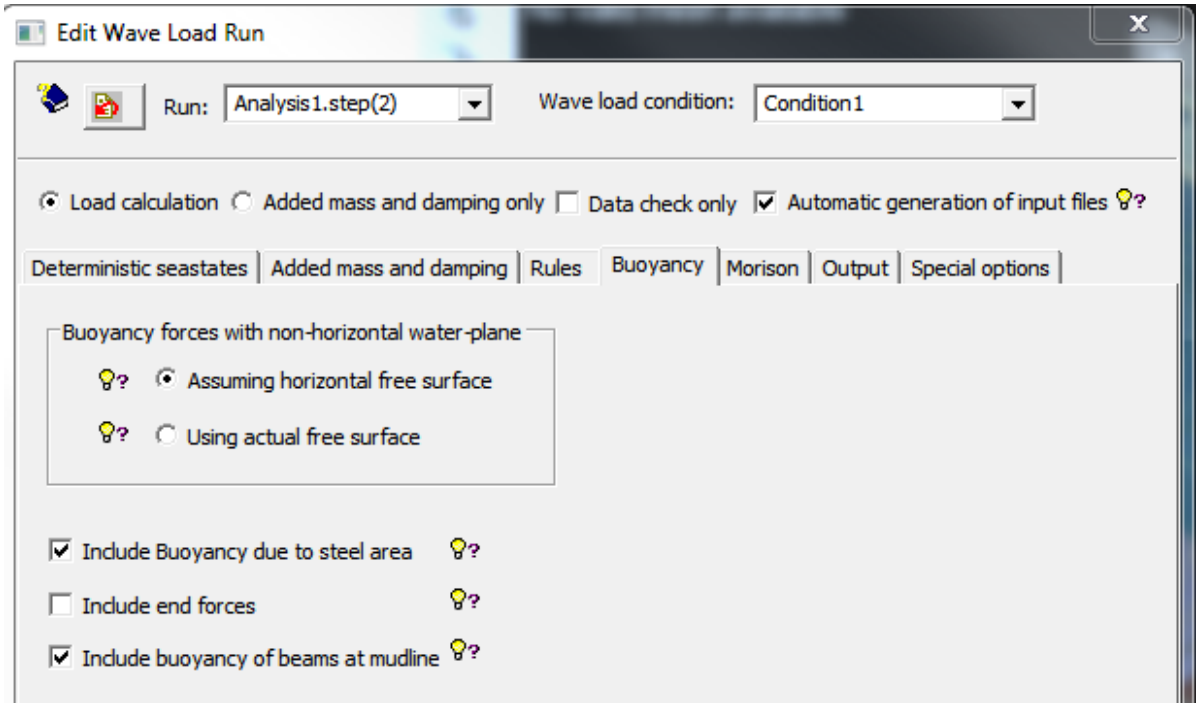


Figure 64: Select the Include buoyancy of beams at mudline option

5.12 Buoyancy area data on concentric (grouted) beams

When importing concentric (grouted) beams the buoyancy area property is defined according to Figure 65.

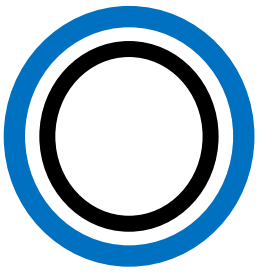
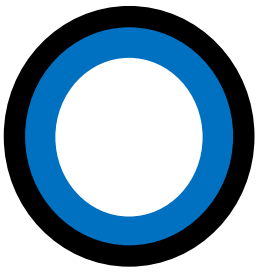
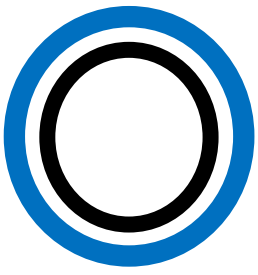
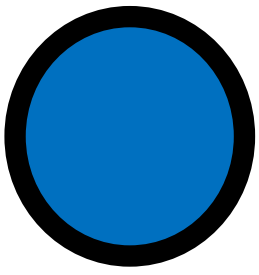
Flooded		**Non-Flooded**	
Outer member	Inner Member	Outer member	Inner Member
Area of steel only	Area of steel + area of grout between inner and outer member	Area of steel only	Area based on radius = Ri, i.e. inner radius of outer member.
			

Figure 65: Illustration of buoyancy area definition of concentric (grouted) beams

Note! Buoyancy area override data from GRPOV or MEMOV card is assigned to the outer beam only. The buoyancy area of the inner beam then equals zero.

Example

The buoyancy areas shown in this example are related to the members using the SECT card of Figure 66. Two beams to the right in Figure 67 have override buoyancy areas based on MEMOV cards in Figure 66.

```
SECT
SECT CO1A      TUB                                  86.361.27  50.8 0.952

MEMOV
MEMOV  20951420           263.488993.147
MEMOV  23042095           263.488993.147
```

Figure 66: SECT and MEMOV cards defining concentric (grouted) members with override

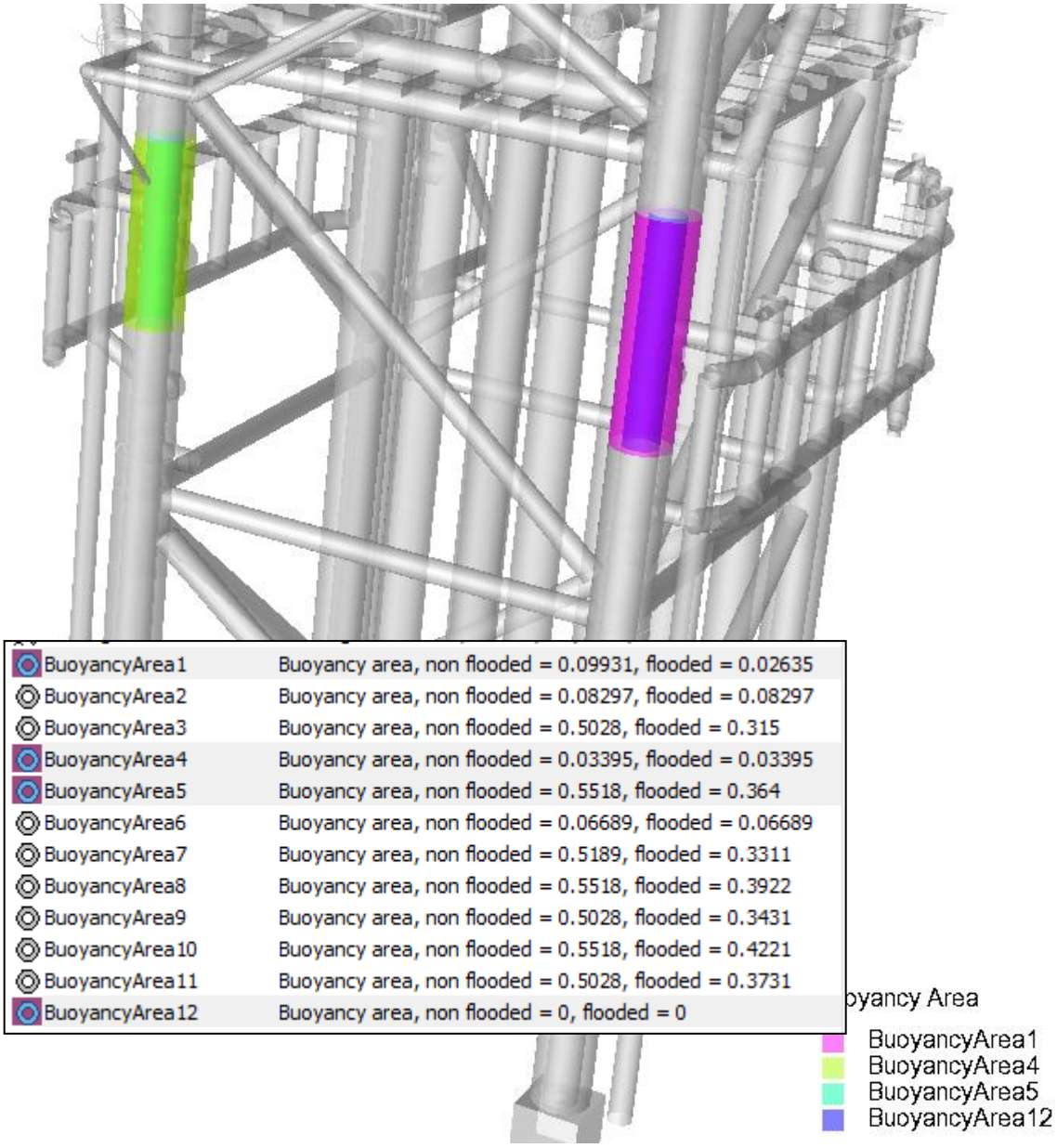


Figure 67: Buoyancy area on concentric (grouted) beams with and without override values

5.13 Limitations in weight import and how to solve them

5.13.1 Load distribution type of surface weight is not supported

The SURFID card of a SACS surface weight contains a “Load distribution type” value that can be <blank>, LX or LY. LX/LY defines that only beams in local X/Y direction within the surface area shall receive load.

The converter converts SACS surface weights disregarding this information and GeniE applies loads on all beams in both X and Y directions, no matter if the weights are supported by beams in X direction only or Y direction only in SACS file. It reduces the load distribution on some beams and could result in un-conservative beam design.

Note that the SacsWeightsToGeniE.js file and Message tab contains a warning when such cards are imported.

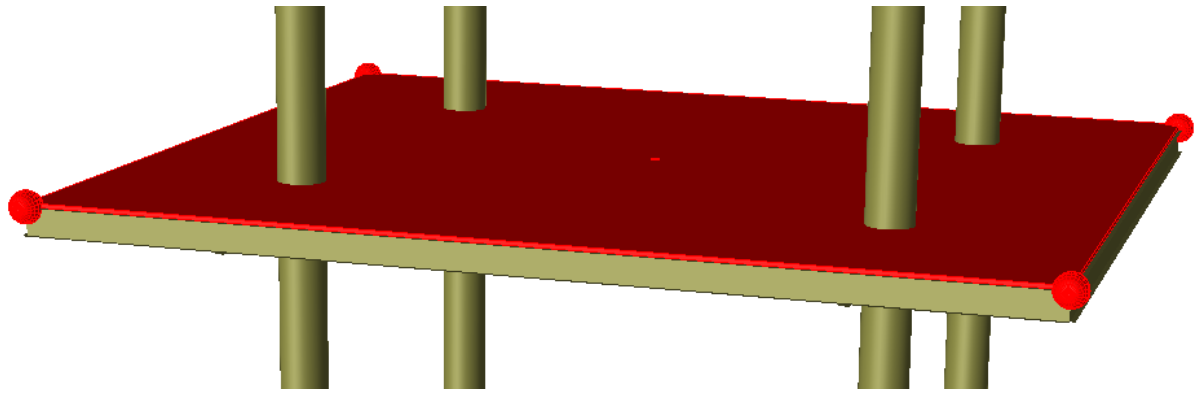


Figure 68: Equipment with footprint that surrounds a regular surface weight area

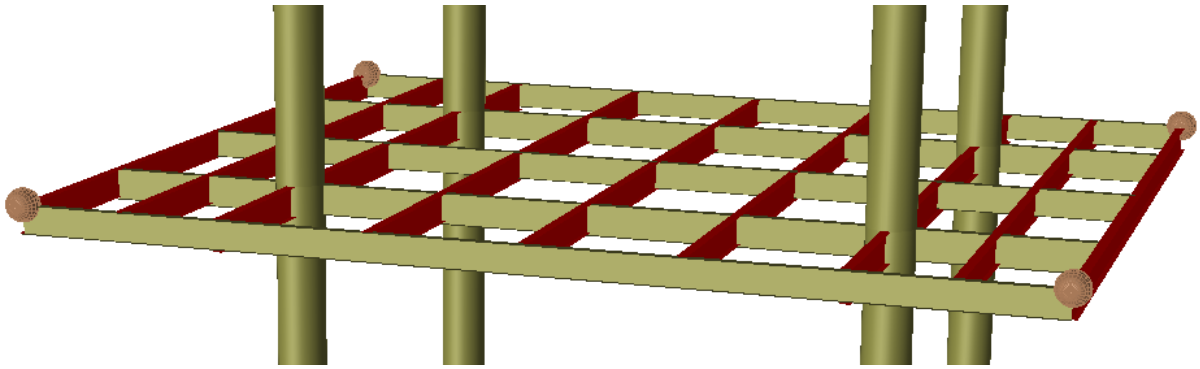


Figure 69: Beams added to load interface assuming load distribution type LY

Workaround

In GeniE, manually define a load interface and add the relevant beams, i.e. beams contained by the boundary joints of the SACS input file and that have the correct direction. Add the equipment to the load interface for relevant load cases. An example of how to select relevant beams is shown in Figure 68 and Figure 69. Note that the boundary joints are shown only to better indicate the area.

5.13.2 Surface weight with irregular area is not properly supported

A SACS surface weight with an irregular area, e.g. trapezoidal or triangular, is partially supported by the converter. The reason is that GeniE only allows footprint areas with rectangular shape. The converter will calculate the total weight and define an equipment with a footprint area surrounding the definition joints as shown in Figure 70.

GeniE applies loads on all beams (in both X and Y directions) contained by the footprint area. It reduces the load distribution on some beams and could result in un-conservative beam design.

Note that the SacsWeightsToGeniE.js file and Message tab contains a warning when such cards are imported.

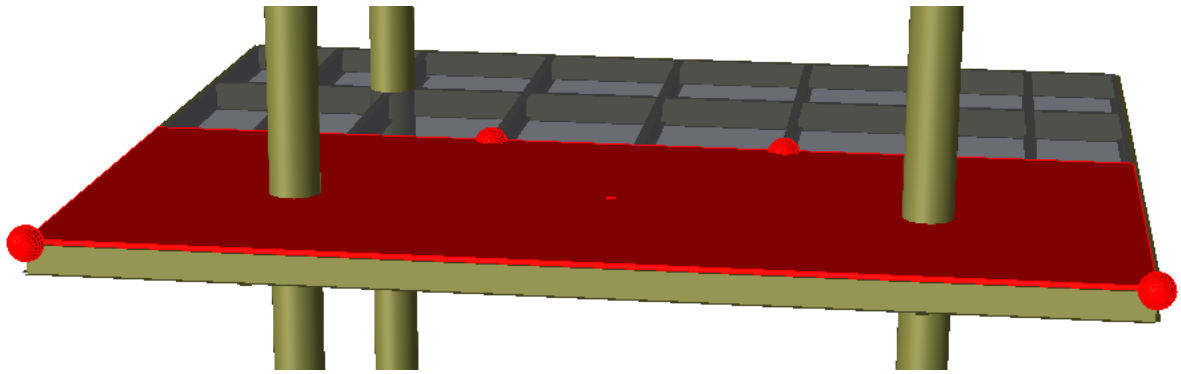


Figure 70: Equipment footprint that surrounds an irregular (trapezoidal shaped) surface weight area

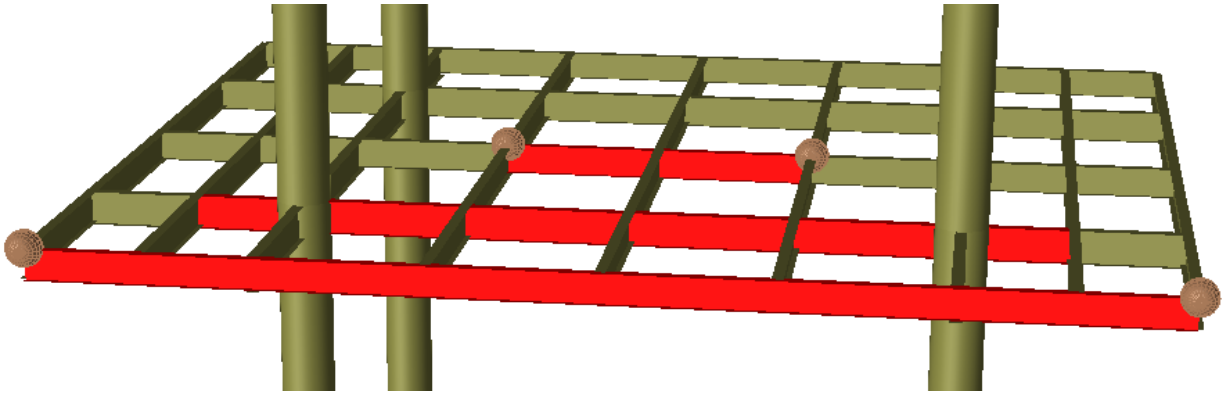


Figure 71: Beams added to load interface assuming load distribution type LX

Workaround

In GeniE, manually define a load interface and add the relevant beams, i.e. beams contained by the boundary joints of the SACS input file and that have the correct direction. Add the equipment to the load interface for relevant load cases. An example of how to select relevant beams is shown in Figure 70 and Figure 71. Note that the boundary joints are shown only to better indicate the area.

5.14 Limitations when GRPOV/MEMOV defined for specific load cases

In a SACS file GRPOV and MEMOV lines may be defined locally, i.e. the group overrides and member overrides will then be applied on groups and members for specific load cases.

In general hydro properties have to be global in GeniE, i.e. they are valid for all load cases. Thus, the converter only imports the set of global GRPOV and/or MEMOV definitions found on the SACS file. If load case specific override definitions exist, the converter issues a warning and ignores them. However, there is one exception. This is described in the following sub-chapter.

5.14.1 Supporting local hydrodynamic coefficients Cd and Cm override values

From GeniE version 7.1 local hydrodynamic coefficients data are supported. The converter will create 'MorisonGlobalDirection' hydro properties when such GRPOV and/or MEMOV definitions are found on the SACS file.

For a more detailed explanation consult the GeniE user manual Vol 2, chapter 5.3.

5.14.2 Supporting hydrodynamic coefficients as factors

If the value of column 77 of the MEMOV/GRPOV card is 'F', the coefficients shall be considered as factors. From GeniE version 7.2 this feature is supported.

For GeniE version 7.1 and earlier no MorisonGlobalDirection properties are generated. A warning message as shown below is written to the Message tab/LOG file.

```
Warning: Unprocessed data on the MEMOV cards. CD and CM overrides as factors is
not supported
```

```
MEMOV  481 20                                        1.171.171.061.06        F
```

Workaround for GeniE version 7.1 and earlier

A possible workaround is to remove the 'F' from column 77 and import factors as coefficients. After the import is completed, the relevant MorisonGlobalDirection properties must be manually updated. The following procedure can used to update the Cd and Cm.

1. Decide the diameter of beams assigned to actual MorisonGlobalDirection property. If several diameters, calculate the average diameter.
2. If the SACS input file contains a CDM card, use the table of diameter dependent hydrodynamic coefficients specified. Otherwise use the following default table.

DIAM.	NORMAL DRAG COEFF.	TANGENTIAL DRAG COEFF.	NORMAL INERTIA COEFF.	TANGENTIAL INERTIA COEFF.
12.0	0.610	0.0	1.39	0.0
24.0	0.665	0.0	1.40	0.0
48.0	0.720	0.0	1.45	0.0
72.0	0.756	0.0	1.60	0.0
96.0	0.781	0.0	1.67	0.0
120.0	0.799	0.0	1.71	0.0

3. If necessary, use linear interpolation to calculate CDN, CDT, CMN and CMT values for the relevant diameter.
4. For each coefficient multiply with the relevant CDN/CDT/CMN/CMT value and update the MorisonGlobalDirection property values accordingly.

5.15 How is soil data converted from SACS PSI input

SESAM Splice program assumes that the soil resistance is constant for each layer. When GeniE converts SACS soil data using the default soil import options the following two scenarios are considered.

1. SACS soil data are constant for a certain layer – GeniE will convert the soil data and place them at the bottom of this layer. The converted data will be used to calculate the constant soil resistance for this layer. If sub layers need to be created, the same soil data will be used for each sub layer.

2. SACS soil data are linearly varying for a certain layer – GeniE will convert the soil data at the top of this layer. The converted data will be placed at the bottom of this layer and used to calculate the constant

resistance for this layer. If new layers need to be created, the soil data at the top of each new layer is calculated based on linear interpolation. The linear interpolated soil data will be placed at the bottom of the new layer and used to calculate the constant soil resistance for each new layer.

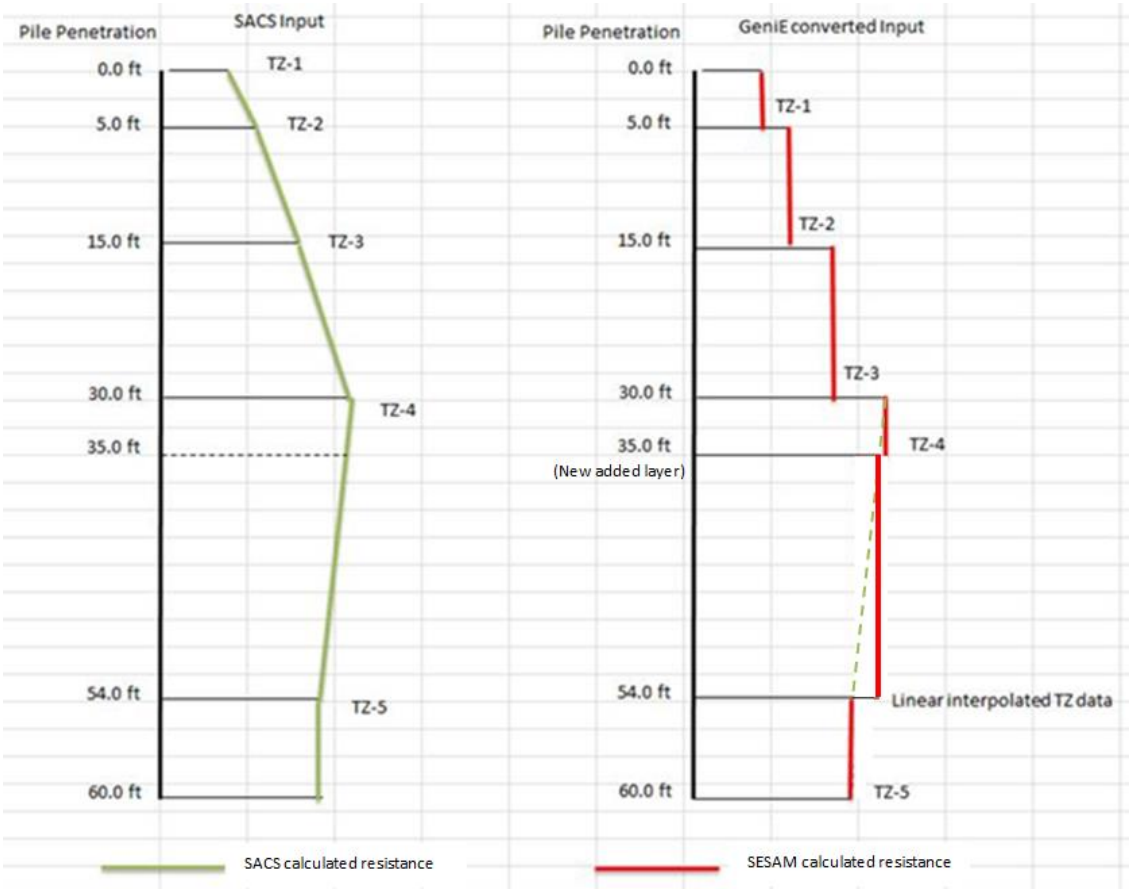


Figure 72: Adding new layers in a linear varying layer (scenario 1)

5.15.1 Refine soil layers

As described above linearly varying SACS soil data is only accounted for in case of new layers being created. By forcing creation of new layers this linear variation may be utilized. This is achieved by using the 'Refine soil layers' options in the Soil tab of the Import SACS dialog:

1. Check 'Refine soil layers'.
2. Add depth and max. layer thickness values at Z1 and Z2.
3. Check 'Make new soil layers.

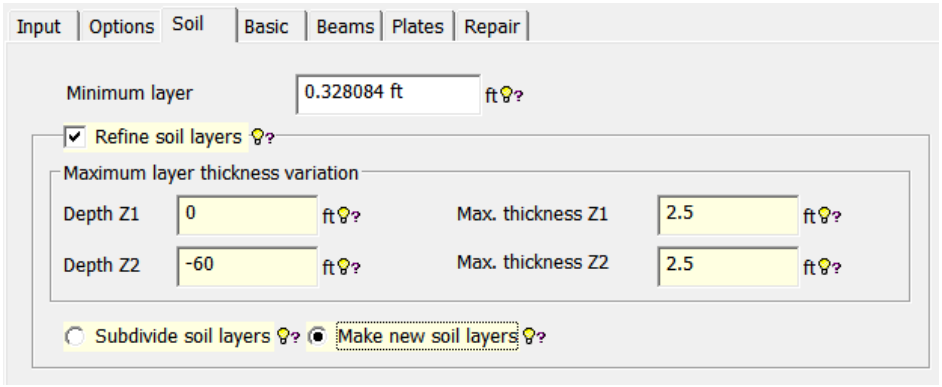


Figure 73: Forcing creation of new soil layers by use of 'Refine soil layers' option

In this case each layer will have 1 sub layer, similar to the behavior using default soil import options described above.

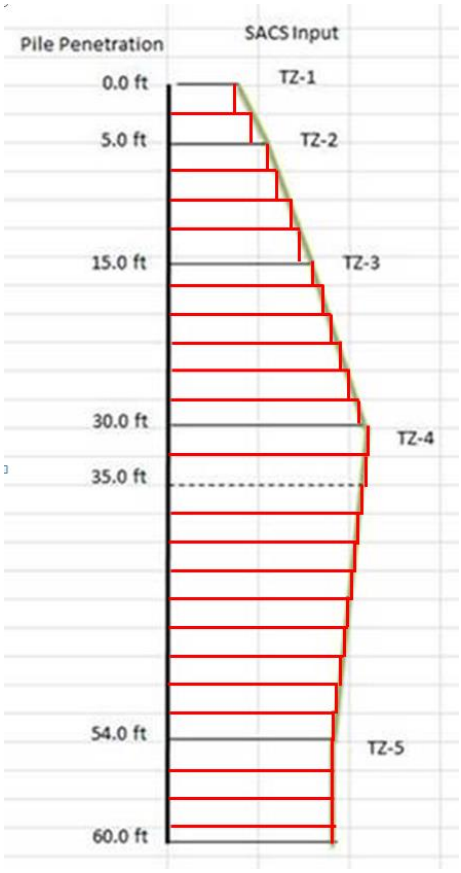


Figure 74: New layers follow the linear variation of SACS soil layers

Note that 'Subdivide soil layers' is the default option when selecting 'Refine soil layers'. In this case sub layers are defined for each layer instead of creating new layers. In this case the linear behavior will not be utilized.

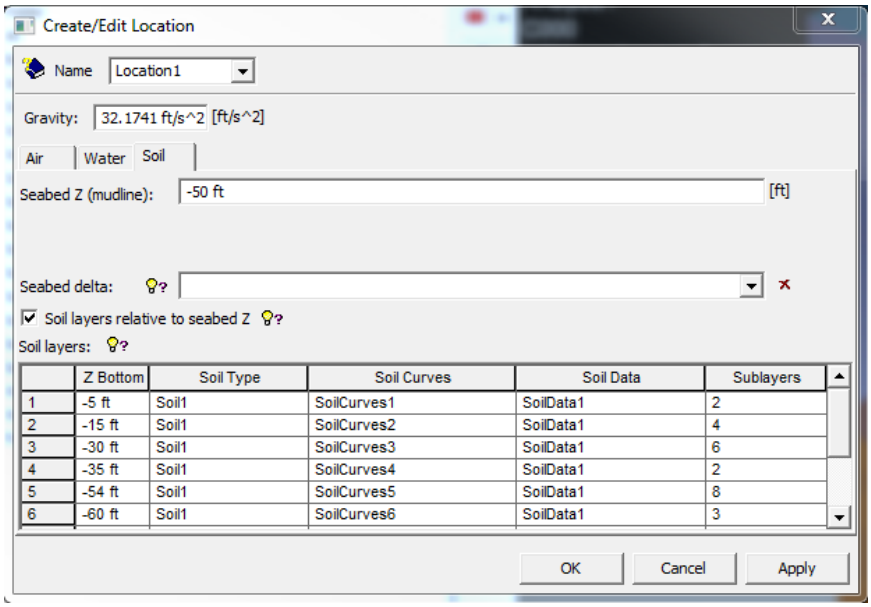


Figure 75: By default 'Soil refinement' define sub layers for each layer

5.16 The GAP_ELEMENT_TYPE sets

In SACS GAP elements can be defined at GRUP lines, MEMBER lines, and/or in a GAP input file. The types of GAP elements defined at GRUP lines and MEMBER lines can be changed in the GAP input file for different load cases, and friction coefficients for friction GAP elements can be defined in the GAP input file. The current converter only supports import of GAP elements defined at GRUP lines and MEMBER lines for all load cases.

MEMBER and/or GRUP cards may define GAP elements of type N, T, C or F. When such elements are imported a set is created for each type. In the table below the set name related to each GAP type is given, along with a comment.

Note that member end releases defined on N, T and C GAP members are skipped. The reason is that truss elements by nature have pinned support at both ends. With regard to non-structural beam elements Sestra requires that both ends are fixed.

Initial gap spacing defined on the MEMB2 card is not supported.

GAP element type	Set name	Comment
N	GAP_TYPE_N	No-load GAP elements converted to non-structural beam elements.
T	GAP_TYPE_T	Tension-only GAP elements converted to tension truss elements which only take tension axial loads.
C	GAP_TYPE_C	Compression-only GAP elements converted to compression truss elements which only take compression axial loads.
F	GAP_TYPE_F	Friction GAP elements are not supported in GeniE and are thus converted to ordinary beam elements. A warning is issued.

Table 2: Relation between GAP element type and Set name in GeniE

The CHK file contains similar information to what is given in Table 2.

5.16.1 Manual actions in case of tension-compression sets

If GAP_TYPE_T and/or GAP_TYPE_C sets are created the user must modify the Analysis in the following way. See also Example A9.

Note that it is allowed to do T/C analysis with wave load data as long as the load output from Wajac is formatted.

1. Select “Tension/Compression Analysis” in the Edit Analysis dialog.

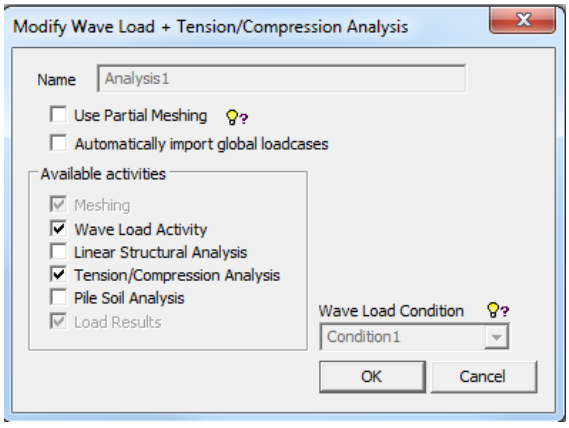


Figure 76: Select "Tension/Compression Analysis" in the Edit Analysis dialog

2. Add loadcases in the Edit Tension/Compression Activity dialog. **Note that the loadcases you include need to be last in the FEM loadcase number sequence of the analysis**.

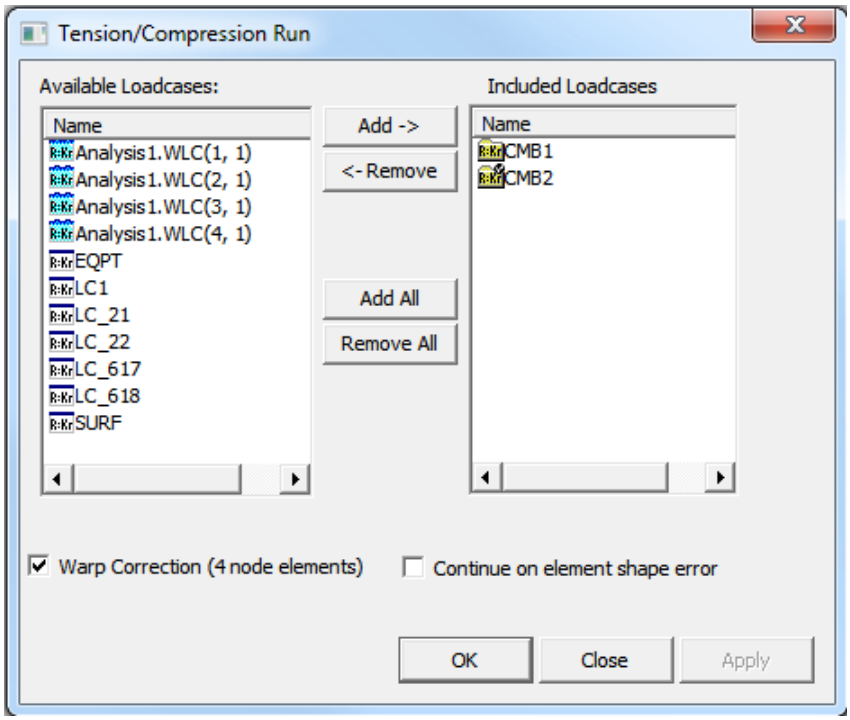


Figure 77: Specify which loadcases that should be run as tension/compression

3. Change load output from Wajac to formatted.

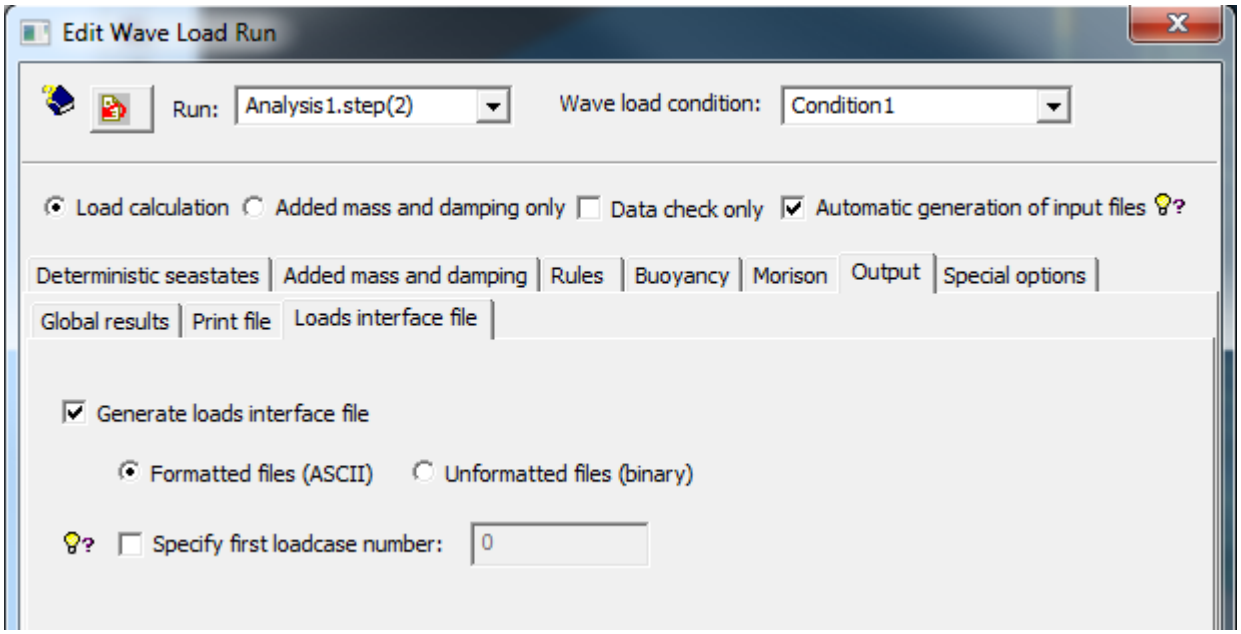


Figure 78: Change load output from Wajac to formatted

5.16.2 Manual actions in case of non-structural sets

See proposals in chapter 5.4.

5.17 Loading condition names with special characters

In GeniE v7.1 and older the following is valid:

Loading condition names with special characters like '+', '-', '.' and '/' are resolved by substituting the special character with '_'. However, in some cases this may result in duplication of loading condition names.

If for example two loading condition names are 'XY+1' and 'XY-1' they will both be resolved to 'XY_1'. This will result in that loads originating from SACS weights/loads are incorrectly imported.

Workaround

Rename loading conditions names on the SACS input file before importing the model to GeniE.

From GeniE v7.2 most special characters are converted to related characters, e.g. '+' to 'P' and '-' to N. Moreover, all conversions will result in unique loading condition names. The LOG file will contain information as shown below.

```
Changed load case names due to special characters:
Original name    New name
------------------------
          PS+         PSP
          PS-         PSN
         -W90        NW90
         W+90        WP90
```

5.18 Limitation in use of cross section area override value

The override cross section area defined in GRPOV/MEMOV cards (col.27-33) is only used to define the buoyancy areas for flooded condition. It does not affect the member self-weight calculations. Note that in SACS this area is also used as the member steel area to calculate the member self-weight.

Workaround

Inspect the SACS input file and check if GRPOV/MEMOV cards holding this information are global only, i.e. not part of any loading conditions. In this case, the relevant sections may be modified in GeniE.

5.19 Limitation in the material thermal expansion coefficient (V7.2 and older)

The thermal expansion coefficient of a material will get the value of 1.2E-5 after import, independent of GeniE database temperature unit.

Workaround

The user should manually verify, and possibly modify, this value of each material property after importing a SACS model to GeniE. If database temperature unit is "delF" (Fahrenheit) the thermal expansion coefficient (ALPHA) of the material should typically be $6.7*10^{-6}$.

5.20 How are loading conditions and load factors converted

From GeniE version 7.1 the conversion of loading conditions is changed. This chapter describes the changes that were implemented for the following reasons

- Separate the calculation of weight of marine growth and buoyancy of total structure

- Utilization of all load factors of the LOADCN card

In SACS a loading condition (named load case in GeniE) is defined by the LOADCN card and may consist of structural and environmental loads and weights. Examples of structural loads are point, line and surface forces/moments given by the LOAD card. Environmental loads are given by cards like WAVE, CURR and WIND, while the DEAD card could represent both structural and environmental loads. Weight is typically given by the INCWGT card.

In addition to this grouping of loads, the LOADCN card carries information of load factors for different types of loads. The following load factors may be given

- Overall → scales all loads of the condition, default 1.0
- Dead → scales dead and buoyancy loads, default 1.0
- Wind/Wave/Current → scales wind, wave and current loads, default 1.0
- User supplied → scales structural loads defined in LOAD lines, default 1.0
- Buoyancy → scales buoyancy loads, default Dead load factor

With GeniE version 7.0 and earlier the LOADCN<name> card resulted in **one load case** with the following characteristics

- load case name is <name>, possibly prefixed with 'LC_' if it starts with an integer
- contains structural loads given by LOAD cards, if any
- includes self-weight if DEAD card exists
- includes Wind Field loads if WIND card representing WIND AREA loads exists
- includes Equipment loads if INCWGT card exists
- defines Design Condition if the global AMOD card refers to the loading condition
- only structural loads are scaled and only by the Overall load factor

Environmental loads, including member wind loads, are represented as wave load cases.

The imported load combinations refer to these load cases and additionally relevant wave load cases resulting from WAVE, CURR and WIND cards.

With GeniE version 7.1 and newer the LOADCN<name> card results in **one load combination** with the following characteristics

- load combination name is 'LCN_<name>'
- contains the load case also defined by older GeniE versions with the same characteristics except for
    - structural loads given by LOAD lines are scaled correctly, i.e. Overall * User supplied
    - self-weight scaled correctly, i.e. Overall * Dead
    - Wind Field loads scaled more correct, Overall * User supplied (~~will be~~ corrected in version 7.2)
    - Equipment loads scaled more correct, Overall * User supplied (~~will be~~ corrected in version 7.2)
- includes a CalmSea state for calculation of weight of marine growth if DEAD card exists
    - scaled correctly, i.e. Overall * Dead

- includes a CalmSea state for calculation of total structure buoyancy if DEAD but no WAVE card exists
    - scaled correctly, i.e. Overall * Buoyancy
- includes a Seastate for calculation of total structure buoyancy if DEAD and WAVE card co-exist
    - scaled correctly, i.e. Overall * Buoyancy
- includes a Seastate for calculation of hydrodynamic forces if WAVE card exists
    - includes wind and/or current profiles if WIND and/or CURR cards exist
    - scaled correctly, i.e. Overall * Wind/Wave/Current

The imported load combinations refer to these load combinations, only. This new behaviour makes chapter 5.10 superfluous.

5.21 Limitations in water depth override

The global water depth is defined on the LDOPT card. However, this water depth may have local overrides on the following cards; WAVE, WIND and DEAD. Currently, only water depth override of the WAVE and DEAD cards are handled.

5.22 Difference in water kinematic viscosity in SACS and GeniE

In SACS the water kinematic viscosity value equals $1.899*10^{-6}$ m^2/s, while the default value in GeniE is $1.19*10^{-6}$ m^2/s. This difference typically represents kinematic viscosity at different water temperatures. The values used by SACS and GeniE are valid for water temperature around 0 and 20 degC, respectively.

5.23 Running not aligned versions of GeniE, Sestra, Wajac and Splice

A particular release of GeniE should be used with the aligned versions of Sestra, Wajac and Splice. Running not aligned versions will most likely lead to that the execution of these programs will fail.

What version of Sestra, Wajac and Splice that should be used together with a particular version of GeniE is from V7.1 found in the Release Notes. Otherwise contact Support at software.support@dnvgl.com.

5.24 The BELOWMDL set

To easily be able to select members where part of the member or the complete member is located below mud-line the named set BELOWMDL is defined.

5.25 Limitations when reading ELASTI definition

The elastic spring to ground definitions defined by ELASTI cards will always be treated as being given in global coordinate system. A warning is written to the .LOG file:

```
Warning: ELASTI with local coordinate system occur, see CHK file. Manual changes
is necessary.
```

Investigate the .CHK file for information about Joint number/name, typically:

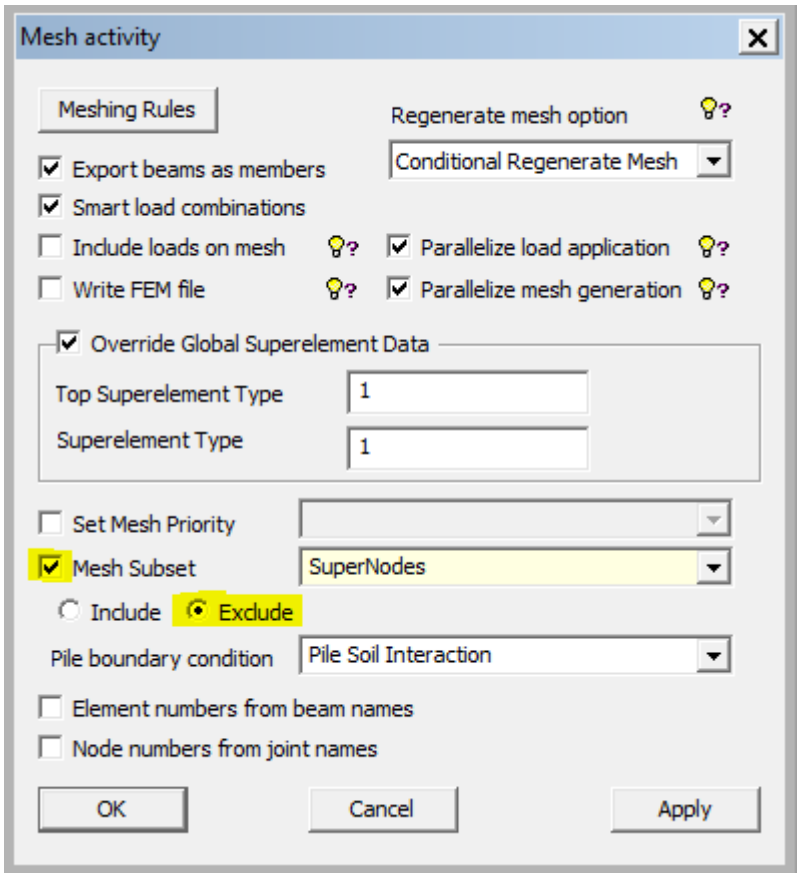
```
Joint xxxx has ELASTI with local coordinate system which must manually be
assigned
```

Also note that when ELASTI and PERSET (PERSET is not converted) are used together, the degree of freedom defined by the PERSET card is converted to a spring with large stiffness.

The user may manually revise the support to include the prescribed displacements for certain load cases after the SACS model is converted.

5.26 Utilizing retained degree of freedom ‘2’ definition

The retained degree of freedom joint support definition ‘2’ is transferred into super-node definition in GeniE. During the conversion these nodes are stored in a named set called “SuperNodes”. After import into GeniE this named set is by default selected to be excluded in the Mesh activity.



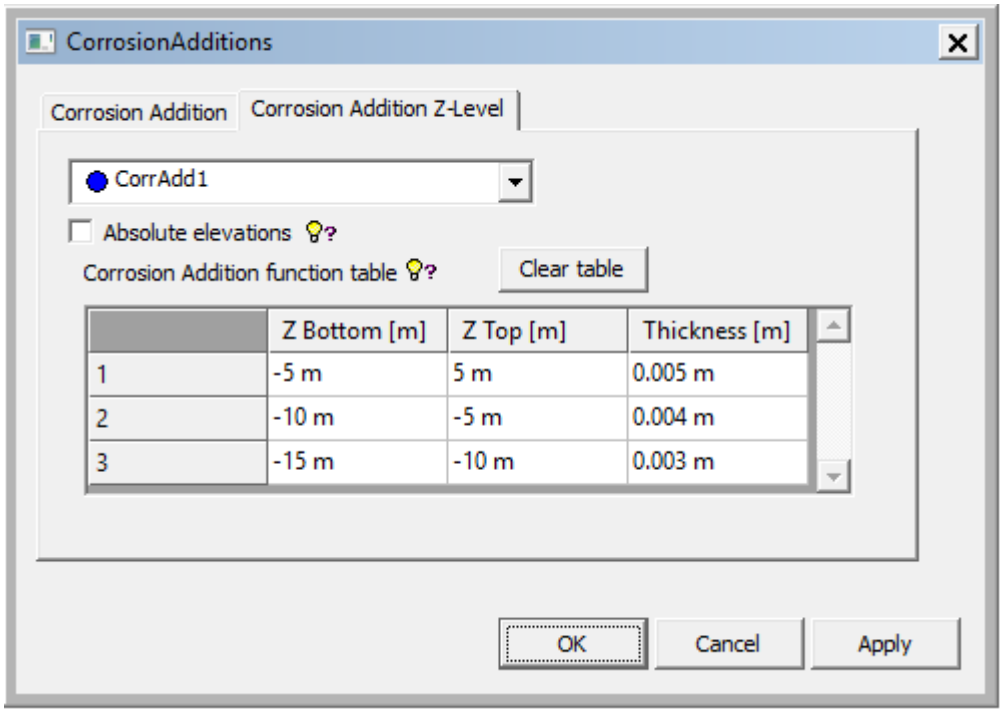
5.27 Limitations in reading CORRZ cards

From v7.14 import of CORRZ cards, with limitations, is supported. The CORRZ card allows the user to specify decreases of section dimensions due to corrosion. The corrosion zones may have gaps but should not overlap. Corrosion zones are defined as a function of elevation and the corrosion thickness may be either constant or linear varying within zones. The corrosion thickness may be specified as an absolute value or as a percentage of the section/plate thickness.

The card has an optional thickness range. By utilizing this range, a zone may define different corrosion thicknesses for different section/plate thicknesses. A minimum corroded thickness and a member loading option can also be specified.

Members that are specified as having no corrosion on the GRPOV or MEMOV cards, will not be modified by CORRZ.

For v7.14 importing corrosion zones with constant corrosion thickness is supported. A Corrosion Addition Z-level property is created and assigned to all beams included in/touching the Z-range of the property. Inner beams are excluded.

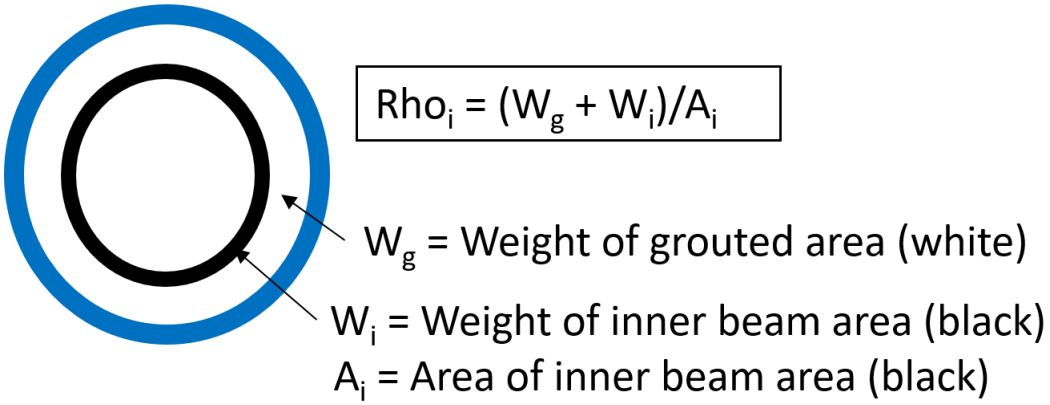


When importing CORRZ cards the following limitations are currently valid.

- Linearly varying corrosion thickness not supported
- Corrosion thickness as percentage of the section/plate thickness not supported
- Optional thickness range is not supported
- Member loading option, use original or corroded diameter in load calculations, not supported

5.28 Equivalent density for concentric (grouted) beams

When importing concentric (grouted) beams the density of the inner beam material is modified as illustrated in the figure below.



Note that in Sacs the density that is used for grout is 150 pound/ft^3. This equals 2.4 tonne/m^3 which is the value used by GeniE when converting from Sacs, i.e. $W_i = Rho_g * A_g$, where Rho_g = 2.4 tonne/m^3 and A_g equals grouted area (white). See also the "Sacs to Sesam" reference document.

6. THE “USE DOMAIN MODEL IMPORT” OPTION

From GeniE V7.3-15 the new FEM (Domain Model) importer may be used in context of SACS import by checking the "Use Domain Model Import" option in the Basic tab of the SACS import dialog. **However, this importer does currently not support hydro properties and code check data. You should only use this option if you experience serious problems of type “Acis error” during import of the structure. All hydrodynamic and code check data needs to be manually inputted into GeniE.**

This chapter describes other issues related to the “Use Domain Model Import” option. The new FEM importer has some features that differ from the old FEM importer. In the following these differences are described.

6.1 Handling of conical transitions

When a beam has a conical transition it is assigned the DefaultCone section property. When this property is assigned to the end segment of the beam it will appear as a line in the graphical window of GeniE. Moreover, Sestra will fail with the following message:

```
  Check of T-file (.FEM) revealed the following errors:

* Geometry data is missing for element    533
  (the reference number in GELREF1 is zero)
```

Note that the “Model Verification” tool will not detect this case as illegal.

Workaround

Select the beam and its neighbours and select “Join…” from the RMB-menu. In the “Join” dialog the “Join only if concepts have same set membership” option must be unchecked.

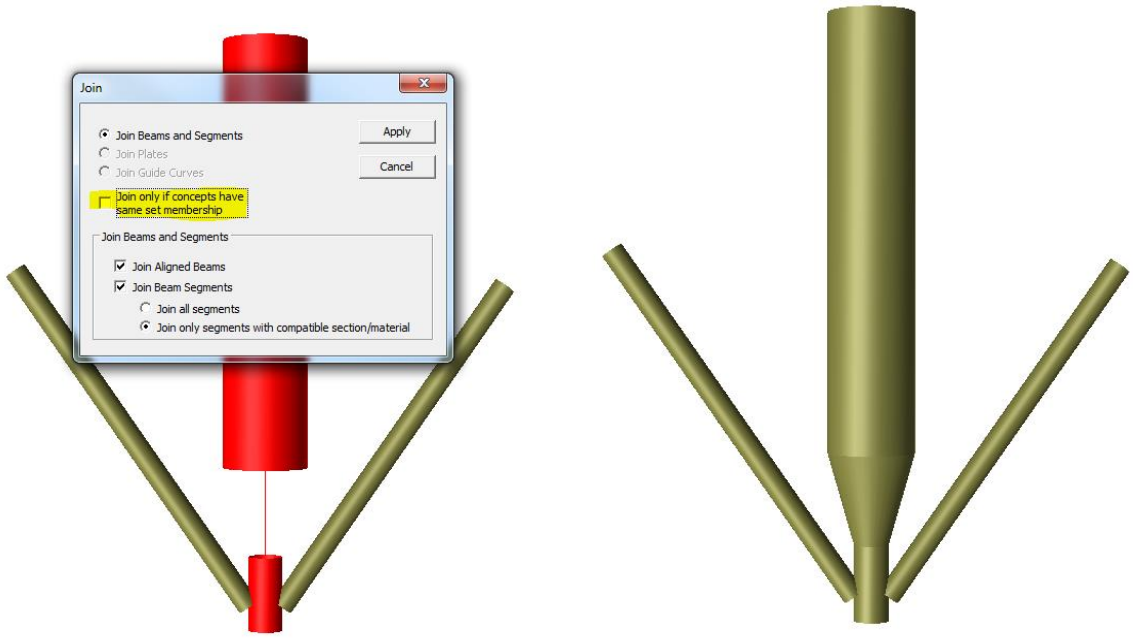


Figure 79: Join beams to make a legal conical transition

Note that the old FEM importer assigns an averaged pipe section to such beams.

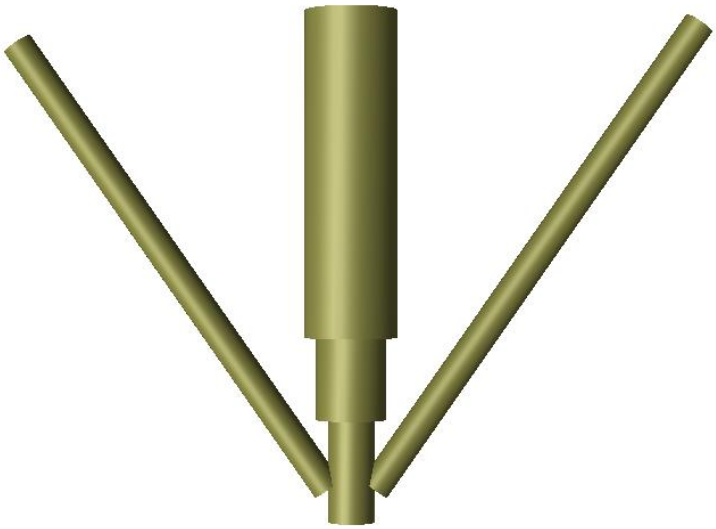


Figure 80: Behavior of the old FEM importer

6.2 Importing plates

When the SACS input contains PLATE cards the resulting plates in GeniE have some different qualities.

6.2.1 Joining of plate elements

The “Allow joining plate elements” option is by default checked when importing a SACS model. How plate elements are joined will vary between the old and the new FEM import, see example below. Typically, the old FEM importer joins more plate elements. In the example below one plate concept per Z-level is made by the old module. This is not achieved by the new module, as can be seen.

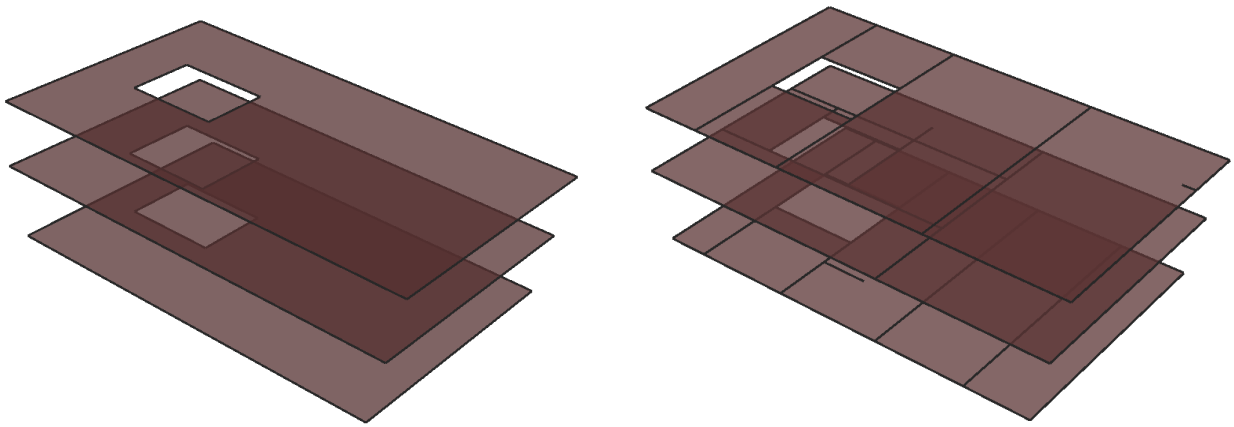


Figure 81: Old and new FEM import module joins plate elements differently

Note that when the “Allow joining plate elements” option is ticked off the result is identical in both cases, i.e. GeniE has the same number of plates as there are PLATE cards in the SACS input file.

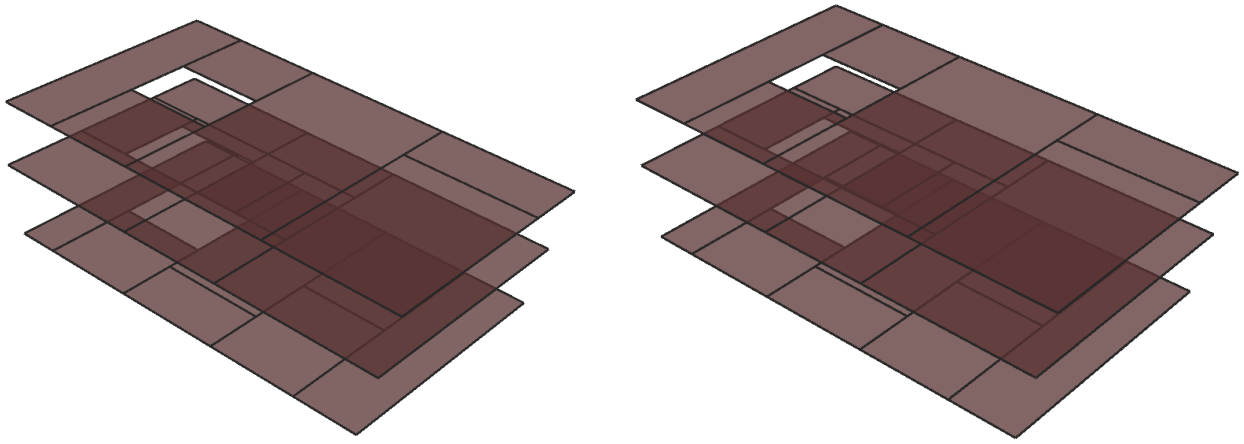


Figure 82: Same number of plates when "Allow joining plate elements" is turned off

6.2.2 Connectivity between plate and overlapping beams

While the old FEM import module will create connectivity between plate and overlapping edges, the new module will only create connectivity at its corners. This is reflected in the mesh and the result is as follows

- When plate elements are connected with beam elements wherever possible all forces are transferred from plates to beams.
- When plate elements are connected to beam elements only at its corners, forces and moments are transferred only at four corners of the plate elements. This simplistic modelling approach may be more in-line with how SACS does the calculations.

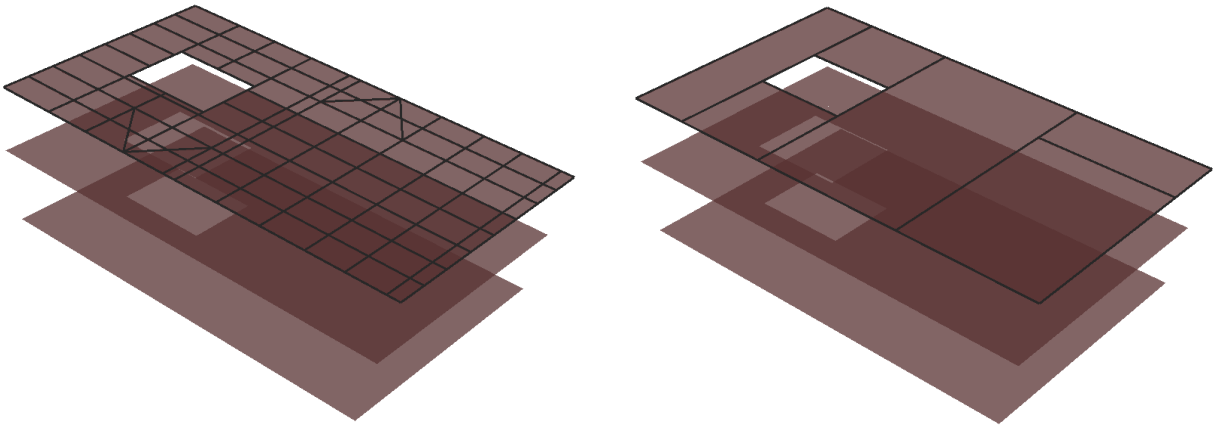


Figure 83: Only the old FEM importer creates connectivity between plate and overlapping beams

Note that in GeniE V7.3 it is not possible to establish the connectivity between plate and overlapping beams when the new FEM importer is used. However, this may be possible in GeniE V7.4.

6.3 Recommended options

Until the hydrodynamic and code check data are handled it is not advisable to select the "Use Domain Model Import" option. If used anyway the following recommendations apply.

If the “Use Domain Model Import” option is checked, it is advisable to uncheck the “Allow joining plate elements” option. This will ensure that there is a one-to-one relationship between PLATE cards in the SACS input file and Plates in GeniE.

6.3.1 Limitations when using model transformation

The “Use Domain Model Import” option activates the possibility to move/transform the model to a new position with respect to the global coordinate system, i.e. to a position different from the joint coordinates on the SACS input file.

However, if importing environmental data and/or pile-soil data the transformation option is locked (greyed out). Model transformation must then be done after import.

You may also use the “old” FEM importer (Domain Model import switched off) in combination with transformation. The same limitations as mentioned above occur. In addition, to open for transformation, you must switch off “Import mesh into analysis”.