

SESAM TUTORIAL

GeniE Import Nastran 3 cargo hold FE model and prepare to apply rule loads from Nauticus Hull

Valid from GeniE 8.12

GeniE WORKSHOP

Import Nastran 3 cargo hold FE model and
prepare to apply rule loads from Nauticus Hull



Sesam Tutorial

GeniE - Import Nastran 3 cargo hold FE model and prepare to apply rule loads from Nauticus Hull

Date: 5 August 2025

Valid from GeniE 8.12

Prepared by DNV – Digital Solutions

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

E-mail sales: software@dnv.com

© DNV AS. All rights reserved

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

Table of contents

1	INTRODUCTION.....	4
2	IMPORT AND CHECK NASTRAN .BDF MODEL	5
3	TEST RUN	10
4	COMPLETE THE MODEL.....	13
4.1	Define yield material data	13
4.2	Compartments, watertight properties and non-structural plates	13
5	BEST PRACTICE.....	15
5.1	Symmetric model	15
5.2	Nastran elements, boundary conditions and MPCs	15
5.3	Prepare sets before exporting Nastran model	16
5.4	Thickness property name update	17
5.5	Section property name update	17
5.6	Load cases	17
5.7	Sesam Converters	17
5.8	FEM import details	18

1 INTRODUCTION

This tutorial shows an example of how to import a **3 cargo hold FE model** from Nastran into GeniE and prepare it for **rule loads** from **Nauticus Hull** for **DNV GL 1A ship** or **Offshore rules**, or **CSR BC&OT**.

A **Nastran** model is exported from FE pre-processors like Patran, Femap and others as a **.bdf** file (bulk data file). If the Nastran file type name is **.dat**, it can usually simply be renamed to **.bdf**, as the format is the same.

Note the following two important **requirements** to the Nastran model:

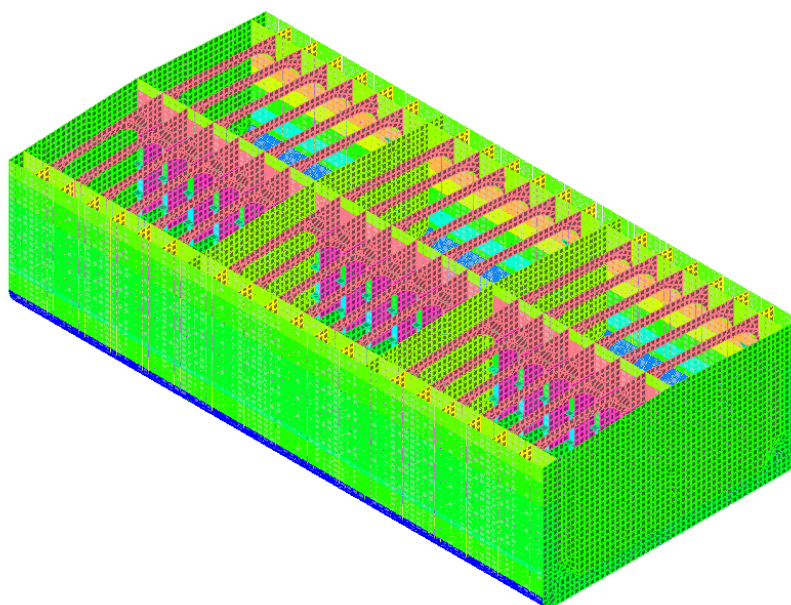
- Beam properties must be exported as **PBARL/PBEAML** cards to transfer sectional data. Cross section data is required for the rule code checking.
- **Gross scantlings** are required. Corrosion additions are added later and differ between FE analysis and code check as well as rule sets.

The Nastran model should have been tested and verified before import, to make sure that the elements are correctly connected, the **mesh quality** is good and that it runs through the Nastran solver. It is possible to use the **FE repair tool** during import (see separate GeniE FE repair **tutorial**), but the easiest and best is to import a good quality FE model.

If unsure about the quality of the model you have received, make sure to use FE repair to check the connectivity and element quality before import.

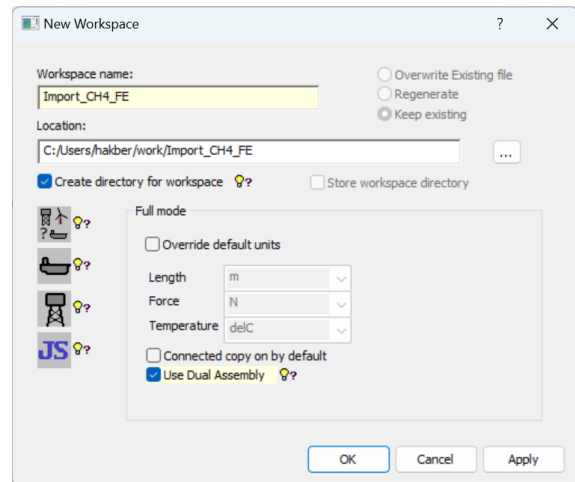
A similar workflow also applies to a FE model created in **Ansys**, except the **.inp** file would have to be converted to Sesam **.FEM** using **Prepost** instead, before importing it as a **.FEM** file into GeniE.

See the last chapter for **tips** and **details** about **sets**, **units**, **Sesam Converters**, **Prepost** etc. See the **Frequently Used Property Sheet** for useful scripting examples.

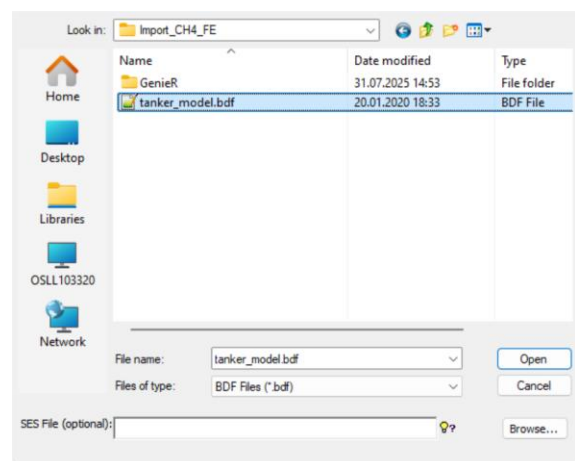
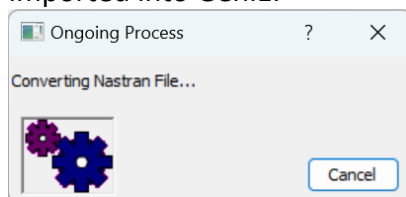


2 IMPORT AND CHECK NASTRAN .BDF MODEL

- Start GeniE and create a new workspace named **Import_CH4_FE**.
 - Check the Use Dual Assembly option (should be used for curved hull models).
 - Optionally click the Ship Mode icon. This will set **mm** as both **Input Units** and **Output Units** for Length.

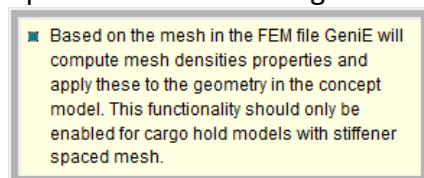


- Open the **File -> Import -> Nastran file** dialog and select the provided **tanker_model.bdf** file. Click Open.
- Sesam Converters will then be run in the background to convert the **Nastran .bdf** file to a **Sesam .FEM** file, which can be imported into GeniE.

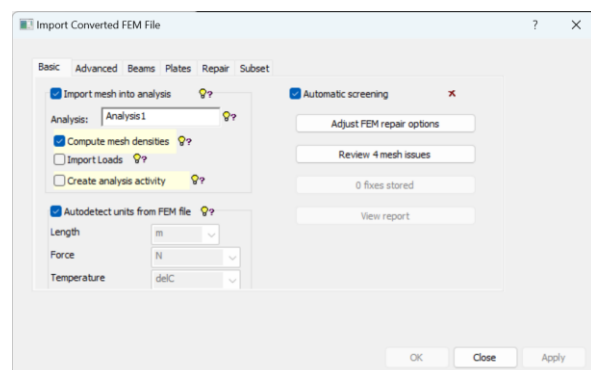


- Note the optional **Patran session file (.ses)** import. It will recreate any **element groups** as sets in GeniE. Note that certain set definitions are **required** later, and it is strongly recommended to create these sets already in the Nastran model. See chapter **5.3 Prepare sets before exporting Nastran model** for details and requirements(!)

- Check the **Compute mesh densities** option in the new dialog:



These mesh densities will help to recreate a similar mesh if the model has to be remeshed.



- Import Loads**

Check to import point and line loads. Surface loads are currently not supported. The

tutorial file doesn't contain any loads.

- **Create analysis activity**

Check this to import the mesh into a new analysis activity.

- **Autodetect units from FEM file**

Will try to detect units based on UNIT card or material properties. If blank, users must manually select units.

- The **Automatic screening** is part of the **FE repair tool** and starts when the .FEM file has been created. The red **x** indicates that mesh issues have been found and the Open button is greyed out.

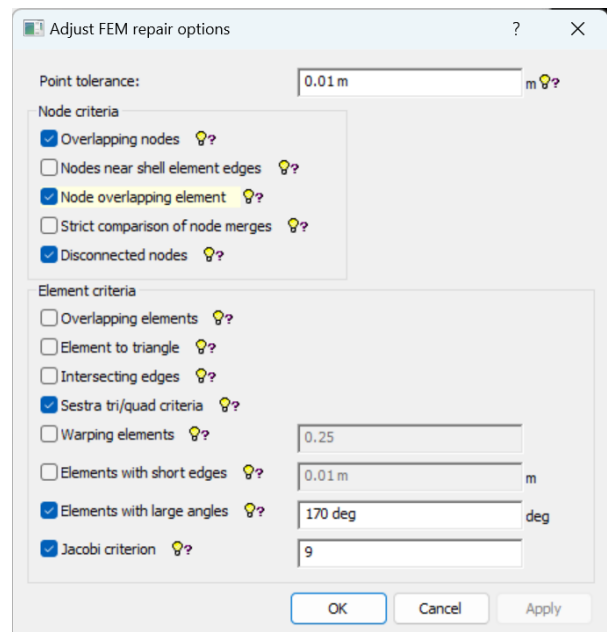
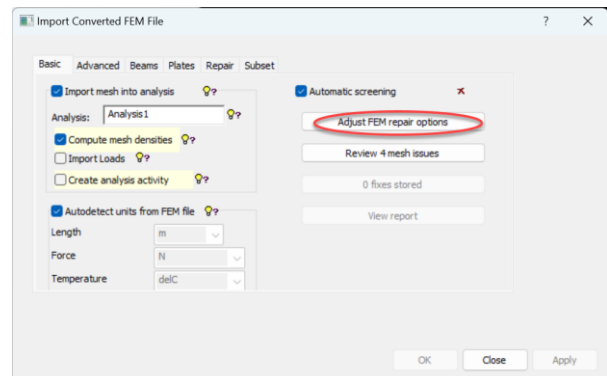
- Open the **Adjust FEM repair options** dialog.

- Look up the Genie User Documentation chapter 15.1.3.8. **FEM Screening and Repair** and read about the different **criteria**.

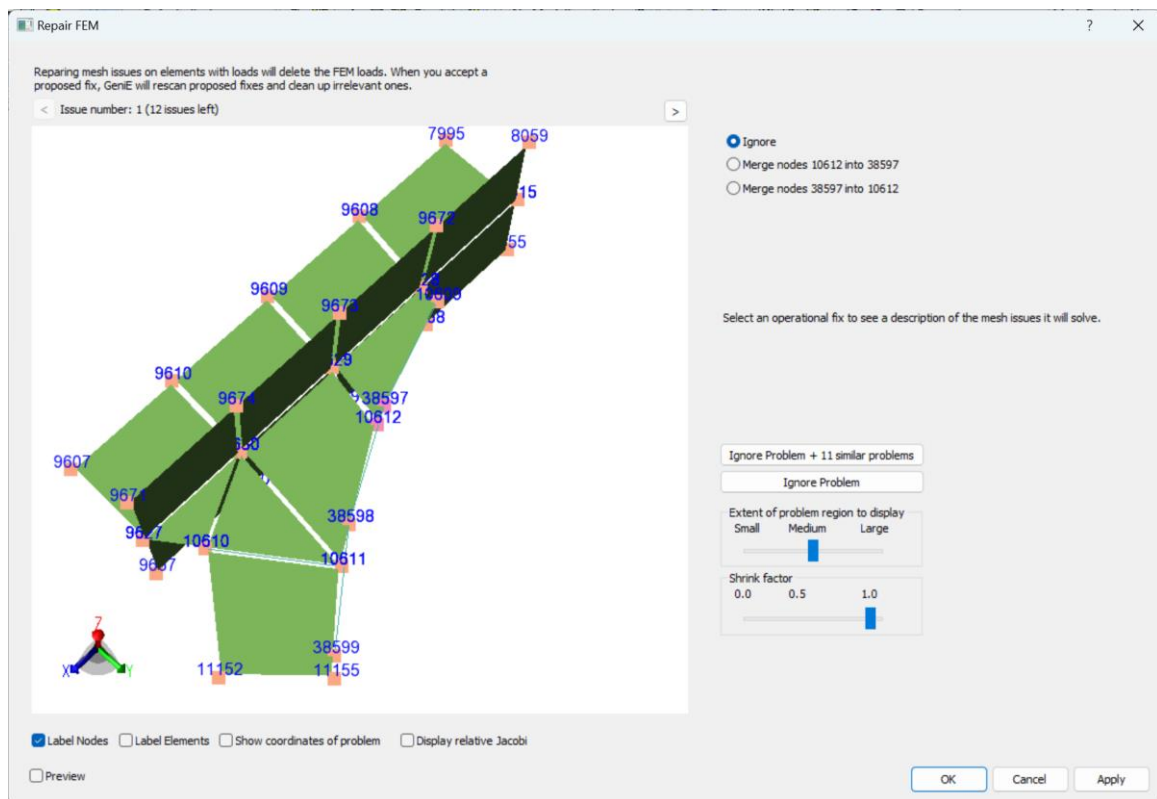
- **Change to selection of criteria** like shown to the right.

- Click OK to close the dialog

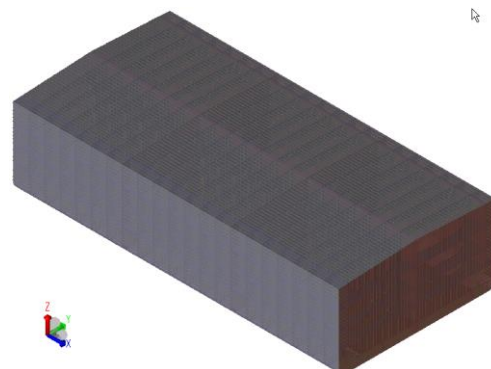
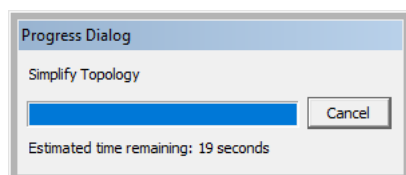
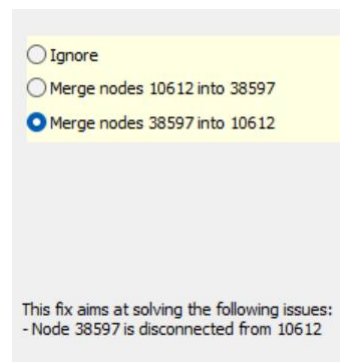
- Details about using this tool is found in a separate **GeniE FEM repair** tutorial.



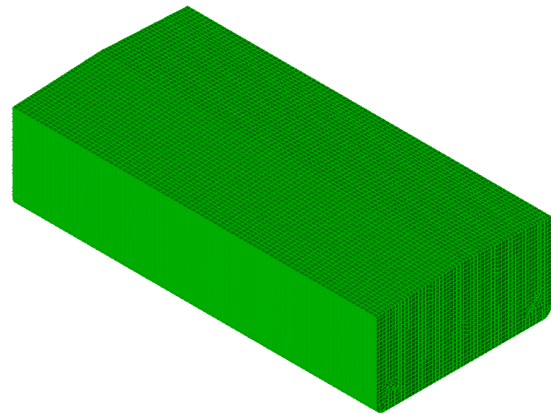
- Click **Review 12 mesh issues** button to open the **Interactive Repair FEM dialog**. It shows some **flange beams not connected to the shell elements**.



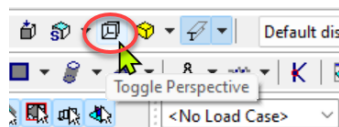
- GeniE classifies mesh topologies around detected problems and can therefore optionally apply the selected fix to (or ignore) all similar problems in the model.
- Selecting a proposed fix shows details about what problem that triggers this proposed fix.
- Select the fix **merging the higher node ids into the lower node ids**.
- Click the button **Accept Fix + 11 similar fixes**, then **OK**.
- Back in the dialog click **OK** to import the FEM file.
- The FE model is imported and a concept model including geometry is created. The original element and node ids are kept.



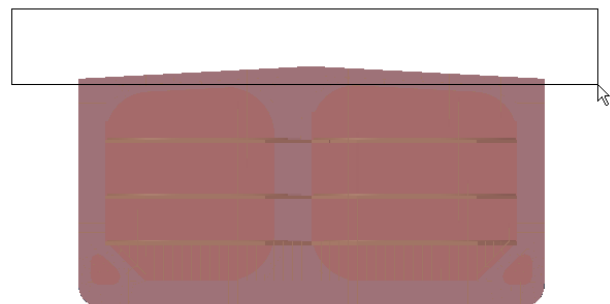
- Change Display to **Mesh – All** and shrink the elements in **View -> Options -> Annotations/Diagrams -> Shrunken Mesh Factor = 0.9**
- Change back to Default display



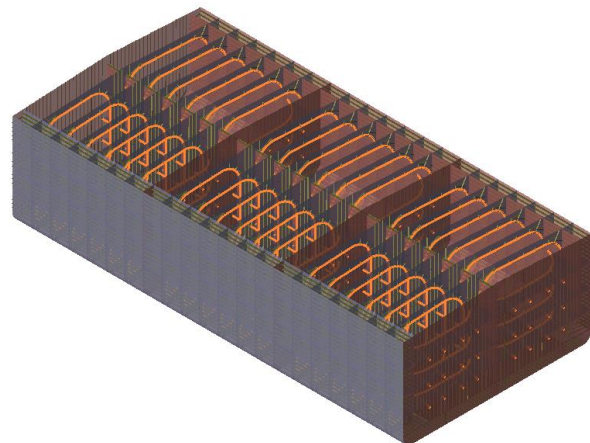
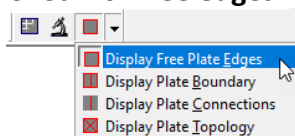
- Uncheck **Toggle Perspective** toolbar icon.

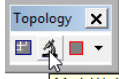


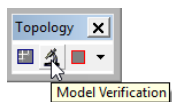
- Press **F6** to view the model from X
- LMB click and drag to select the strength deck, then press **Alt+Minus**.
- Press **F5** for Iso view

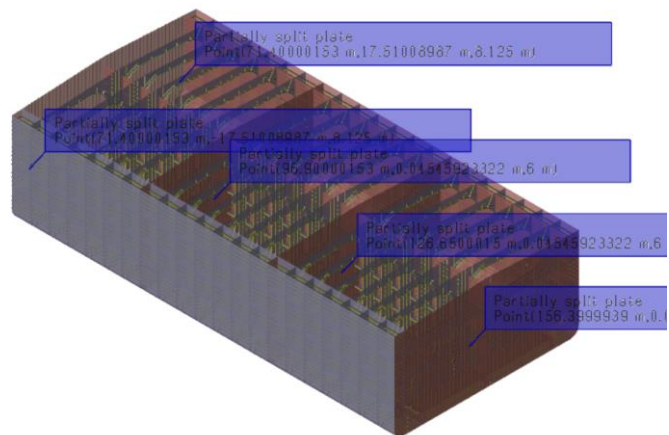
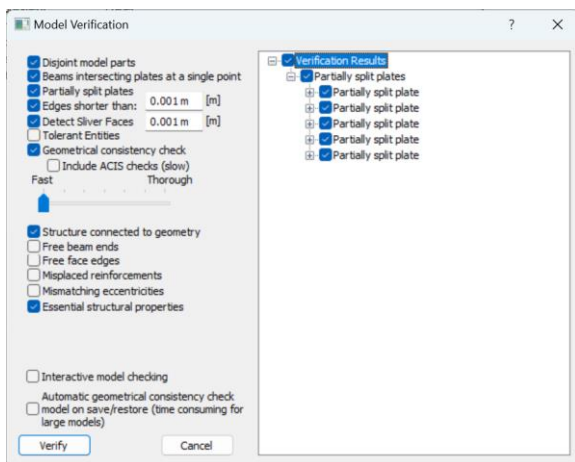


- Check for **free edges**

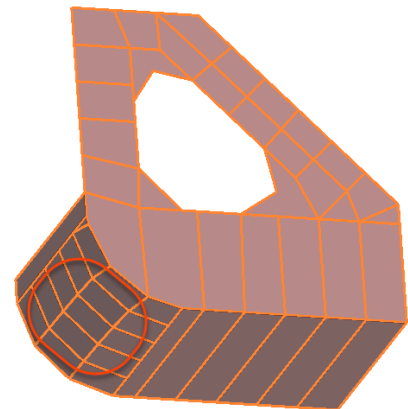


- Click the **Model Verification** toolbar icon  then **Verify**

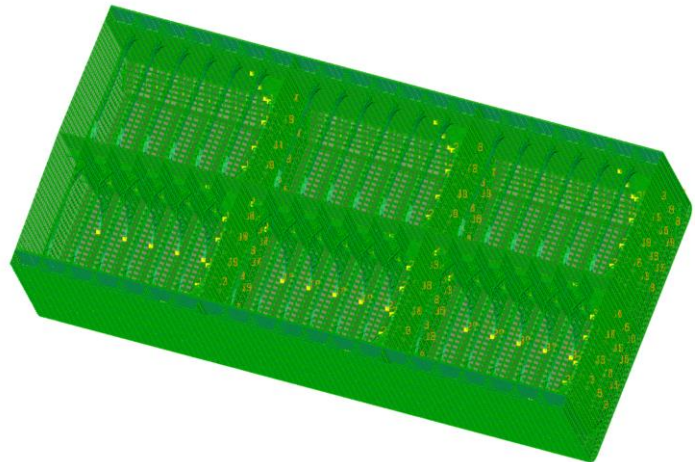
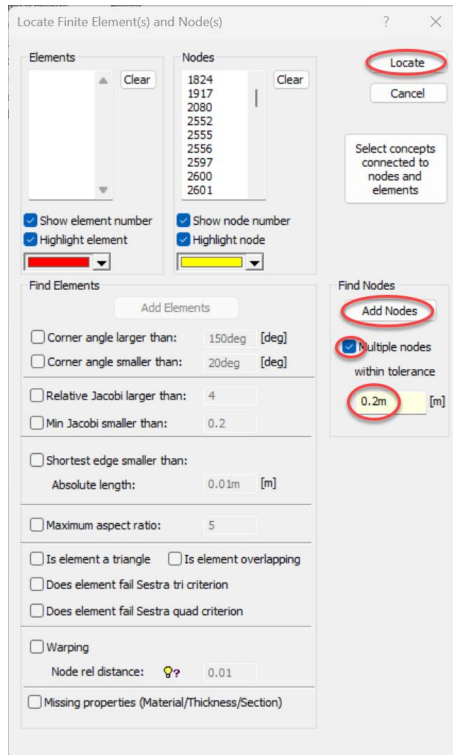




- Always zoom in and take a closer look at the reported results, try to understand them and address any problems. In this model the partially split plates were originally split on purpose or they are due to a couple of originally missing beam elements.
- Ask the instructor to show how to review the above results effectively.
- Note that for curved shells the imported mesh edges are currently persistent internal edges that can't be simplified away, whilst the mesh rule **Simplify Topology** works as usual for imported flat plates. Hence, curved shells may have to be recreated if a different curved mesh is needed.
- The marked edges reflect the imported element edges, and any regenerated mesh will follow this pattern. A finer mesh (further subdivision) may still be created. On the attached flat plates, only the beam-plate intersections remain after simplify topology.



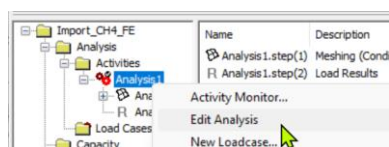
- Change display to **Mesh** – **Transparent** and open the **Mesh & Analysis -> Locate Finite Element** dialog and find nodes within 200 mm tolerance:



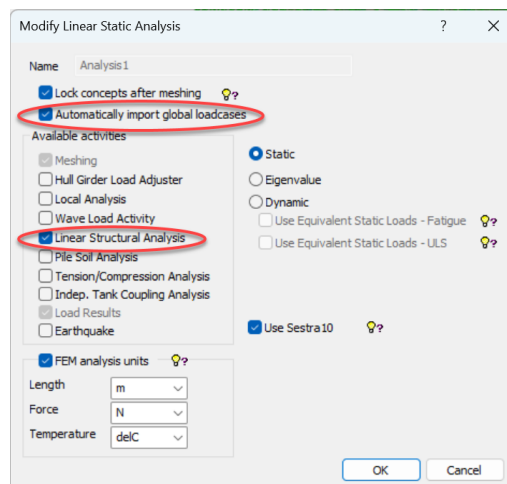
- Click **Add Nodes** before **Locate**.
- Zoom in and check the nodes found. The tolerance of 200 mm finds many bracket toes, that can be left as-is.
- Click **Clear** button and **Locate** again to remove the labels.

3 TEST RUN

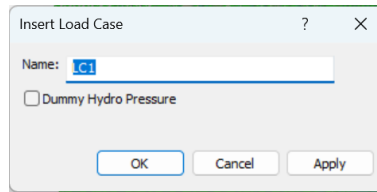
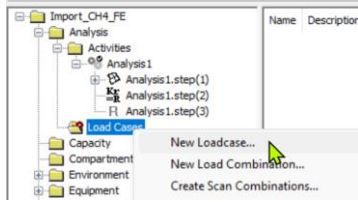
- Open **Edit Analysis** dialog and check **Automatically import global loadcases** and **Linear Structural Analysis**



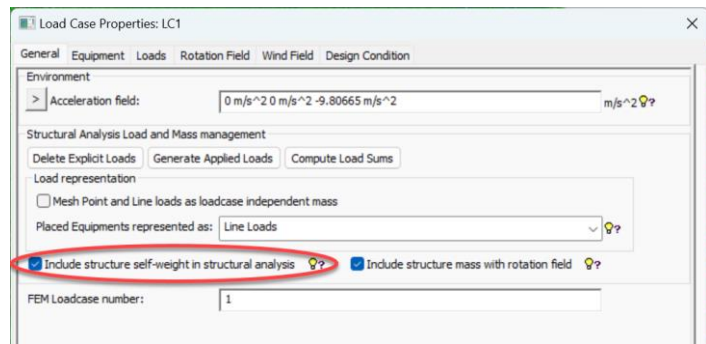
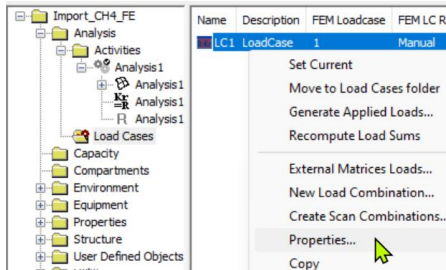
- Click **OK**



- Insert new loadcase, click **OK**

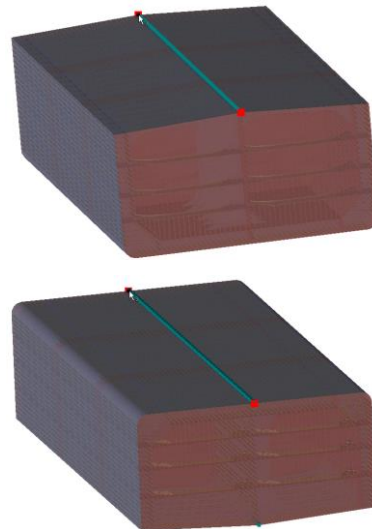
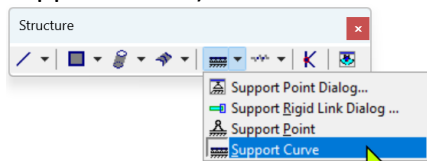


- Select loadcase, RMB click and open properties dialog

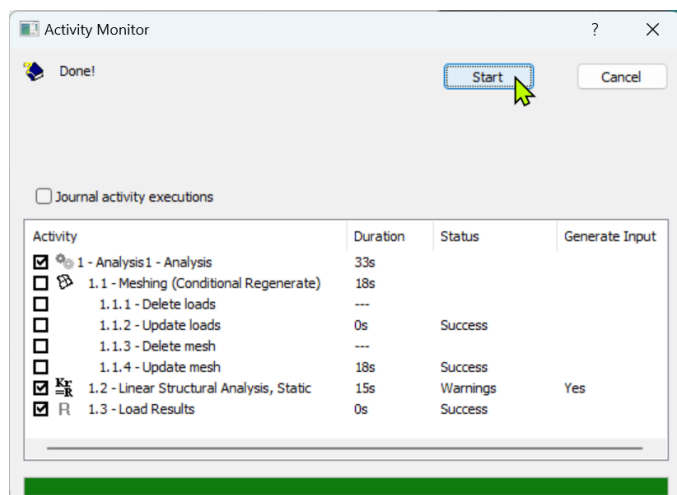


- Check **Include structure self-weight in structural analysis**

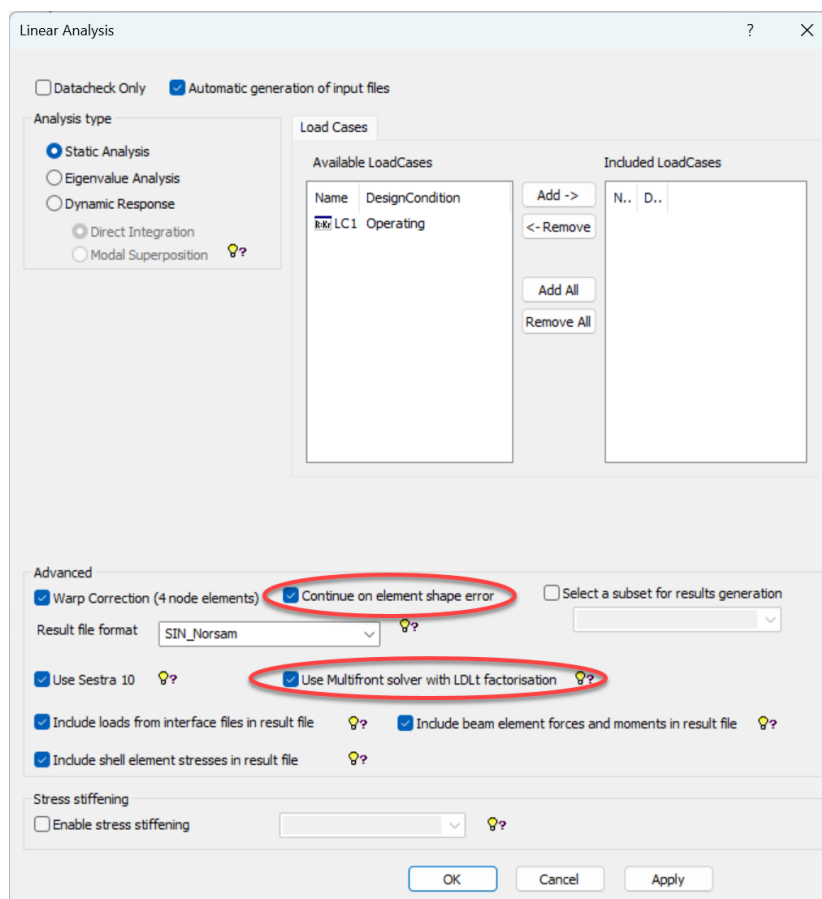
- Add two support curves along the centre line, on top and bottom. (Click the end point twice to create the support curve.)



- Press **Alt+D** then **Start** to run the analysis



- **Always check the and sestra.mlg file for warnings and errors!** Address any issues.
- **Check that correct units** are used.
- **Check load sums, mass and COG.** Compare output from Nastran .f06 and Sesam Sestra.lis files.
- Do some spot checks of beam orientations and eccentricities
- A **recommended model check** is to apply 1G loading in x-, y- and z-direction in separate load cases and check the displacements, or also run an eigenvalue analysis and check that the first modes are global modes.
- Enabling the following options in the **Linear Analysis** dialog may allow a mesh of less quality through the solver:



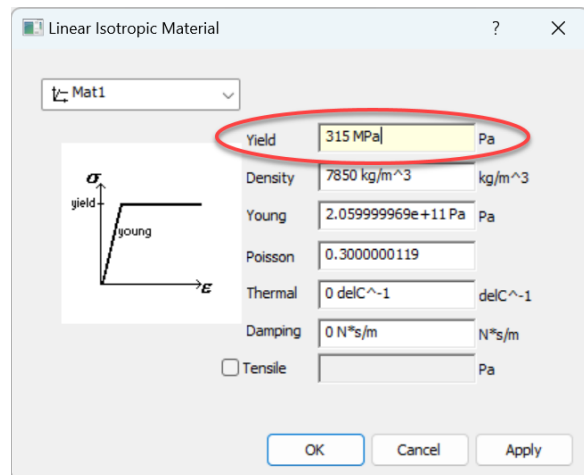
- Check the results properly when using these options, especially if negative eigenvalues are reported for the alternative factorization method. A good model should pass with default settings.

4 COMPLETE THE MODEL

4.1 Define yield material data

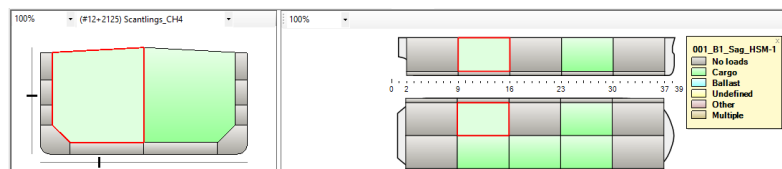
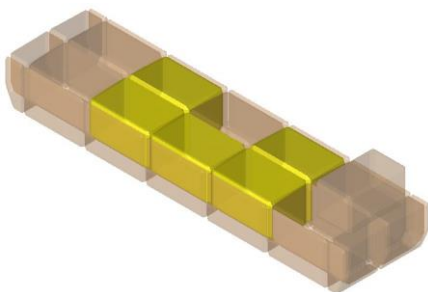
- Input material data for yield if not defined. This is required for any code checks.

Double-bottom

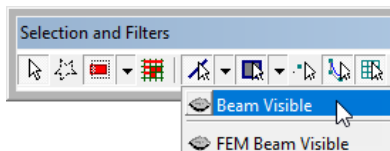


4.2 Compartments, watertight properties and non-structural plates

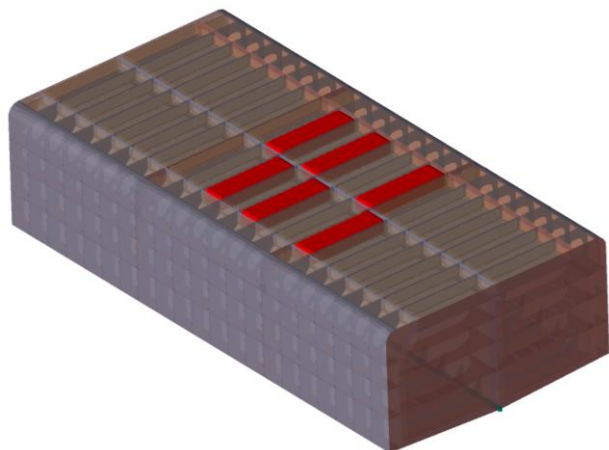
- Click **Loads -> Compartment -> Compartment Manager** to insert compartments manager. A compartment is then automatically created for each closed volume in the model. To apply rule loading from Nauticus Hull later, the **GeniE compartments must match the compartments in Nauticus Hull**.



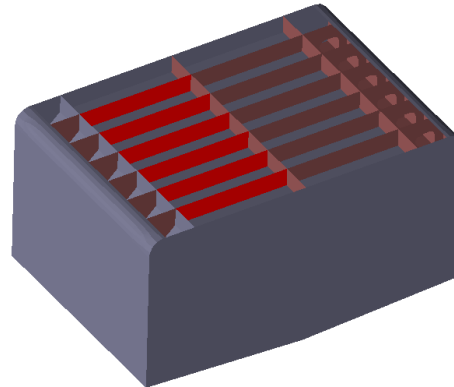
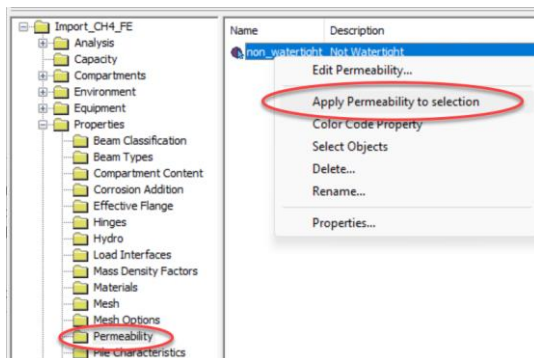
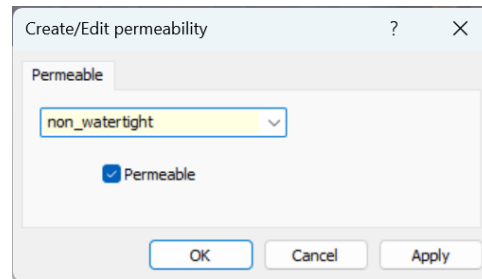
- All plates are by **default watertight** unless they have holes or cut-outs.
- Hide all beams by **RMB** clicking the beam selection toolbar icon and close the eye:



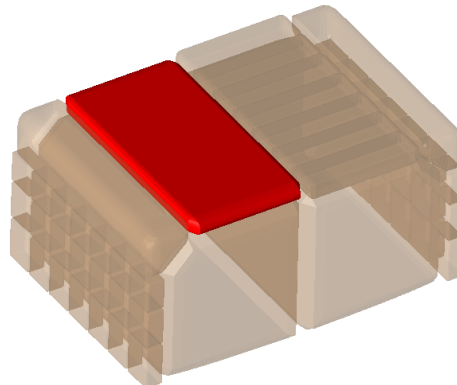
- Hiding the outer bottom plates and selecting a few compartments, it is seen how the webframes divide the ballast tank into multiple compartments.



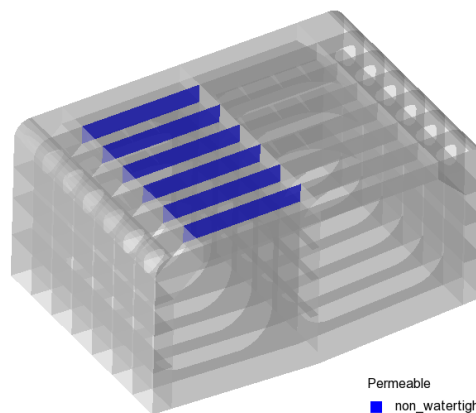
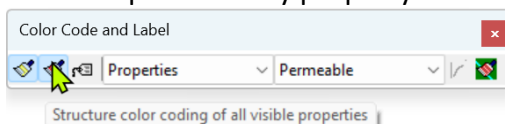
- To merge compartments a **permeability property** must be assigned to the plate between the compartments.
- **Edit -> Properties -> Permeable tab -> Create/Edit Permeable**
- Name it **non-watertight** and click OK
- Select the webframe plates in the double-bottom and assign them the new property



- Set display to **Compartments**
- See how the small compartments have been merged across the plates.

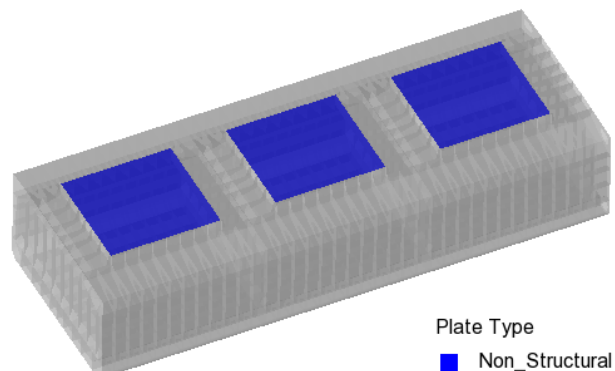
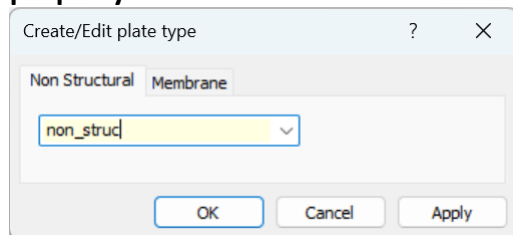


- Switch back to Default Display and color code the permeability property



Permeable
■ non_watertight

- If there are **openings that need to be closed to form a compartment**, for example non-modelled hatch covers on a bulk carrier, the opening should be closed with a normal plate and then assigned a **non-structural plate property**.



5 BEST PRACTICE

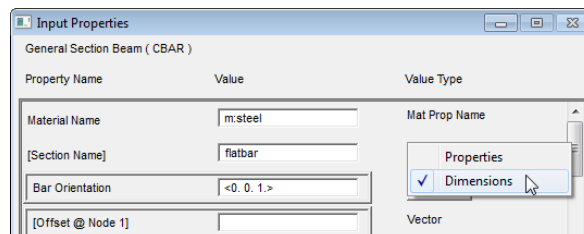
5.1 Symmetric model

Mirror symmetric models in Nastran before export.

5.2 Nastran elements, boundary conditions and MPCs

Beam properties should be exported as PBARL/PBEAML cards to transfer sectional data. Without this, the geometry of the cross sections can't be recreated in Sesam. Cross section data is needed for code checking.

In MSC Patran this is done by exporting beam properties as dimensions instead of properties



Note that:

- **CBEAM** elements should be used instead of CBAR elements in Nastran if the shear centre axis is not coincident to the neutral axis. The converter should warn if this has not been followed. PBEAML should be used instead of PBAR/PBARL for the following Nastran sections types 'I', 'CHAN', 'T', 'CHAN1', 'T1', 'CHAN2', 'T2', 'L' and 'BOX1'.
- Nastran CROD elements have axial and torsional stiffness whilst Sesam TESS(10) truss elements have only axial stiffness.
- Tapered beams are not supported by Sesam Converters
- Solid elements are not supported by Sesam Converters/GeniE

It is convenient to prepare **boundary conditions** (SPCs) at the ends of the model in Nastran and import into GeniE. These can be used when creating support rigid links in GeniE later.

Multi point constraints (**MPCs**) should be **recreated** in GeniE. They can be deleted before export to Nastran, manually in the .bdf or .FEM file or after import into GeniE. RBE2 is generally supported but other MPCs are not. Note that there are slight differences in the behaviour between the MPCs in Nastran and linear couplings in Sesam.

5.3 Prepare sets before exporting Nastran model

Note that **elements group/set/layer** information is not supported by the bdf file format. Sesam Converters can however convert such information from a Patran session file format to Sesam sets. **MSC Patran and Apex, Femap, FEGate, KR Hullscan and HyperWorks can export a Patran session file** with element groups information:

- Patran: Utilities -> Group -> Groups to Session File
- Femap: Custom Tools / Grouping / Export Groups to Patran Session File
- FEGate: Manage Group file option -> Export -> format = PATRAN session file
- KR Hullscan: Menu -> Export -> Export Group Data
- Abaqus CAE: File -> Export -> Patran Session File (.ses)
- HyperWorks: Set Panel -> Export -> Patran session file
- Ansys: No available option. Use scripting or other tools to export a file on the format shown below.

Transferring group/set/layer information not only saves recreating the sets in GeniE, but also acts as **boundaries** for joining elements during import. It is **recommended to make separate sets** for:

- Each cargo hold
- Each compartment
- Evaluation area (usually -1 web ~ +1 web from target hold)
- hgla_hgbe and hgla_hgse sets for hull girder load adjustment
- All members from bottom deck to 1.5m above for container hold (for corrosion requirements)
- The area within 3m above bottom and below top for separate corrosion addition requirements (CSR BC&OT)
- Wet surfaces for outer shell and deck (required for ruleloads import of aftmost or foremost models, but not midship models)
- Sets for permeability properties, non-structural plates (typically hatch covers) and nodes for boundary conditions
- Toes of web frames/stringers
- Grouping elements for every deck in E/R area and fore-body area
- Buckling type method, i.e. SP-A, SP-B, UP-A, UP-B (DNV, CSR BC OT Rule)

All regular sets for transverse webframes, bulkheads, decks etc are part of the ruleloads transfer

from Nauticus Hull.

Furthermore, note that all boundaries of compartments must be closed using dummy elements (non-structural plate in GeniE), i.e. pipe duct or passageway of container ships, for compartment creation. If boundaries are not closed, the volume is not closed and the compartment is not recognized/created in GeniE. If there is no compartment, the corrosion will not be applied correctly. These non-structural plates can be created in GeniE, but it may be convenient to create elements for them in the model before export to Sesam.

The Patran session file format looks like:

```
-----
ga_group_create( "58_Transverse stringer 6000 aBL" )
ga_group_entity_add( "58_Transverse stringer 6000 aBL", @
" Node 1642 1770 1772 1899 1901 1929:1937 1940:1945 1947:1948 1955 1958:19" // @
"68 1972:1973 1975:1980 1988 1990:1993 1995:1999 2001 2023 2025:2030 2034 " // @
"2036 2038:2040 2042 2045:2050 2053:2054 2056:2057 2059:2063 2066:2070 207" // @
"2:2079 2081:2084 2088 2090:2091 2093:2096 2099 2101:2104 2106:2111 2116 2" // @
"121 2137 2167:2182 2184:2189 2192:2202 Element 30813:30962" )
ga_group_create( "59_Transverse stringer 11600 aB" )
ga_group_entity_add( "59_Transverse stringer 11600 aB", @
" Node 1630 1678 1712 1833 1865 1867 2206:2228 2230:2252 2275:2287 2295 22" // @
"97 2299 2301:2303 2305:2306 2309 2311:2312 2314:2315 2317:2318 2320:2321 " // @
"2323:2324 2327:2328 2330:2331 2335 2338 2341 2343:2345 2347:2349 2352 235" // @
"4:2356 2358 2360:2361 2364 2366:2368 2371 2373:2375 2377 2379:2381 2383 2" // @
"385:2388 2390:2392 2394:2395 2397 2399:2404 2406:2411 2413:2419 2427 2429" // @
":2432 2435 Element 30963:31112" )
sys_poll_option( 0 )
```

5.4 Thickness property name update

The script **rename_thickness.js** provided with this workshop can be run to loop over and rename all thickness properties to reflect the actual thickness value.

5.5 Section property name update

The script **rename_section.js** provided with this workshop can be run to loop over and rename Bar, L Section, Unsymmetric and I Section properties to reflect the actual section type.

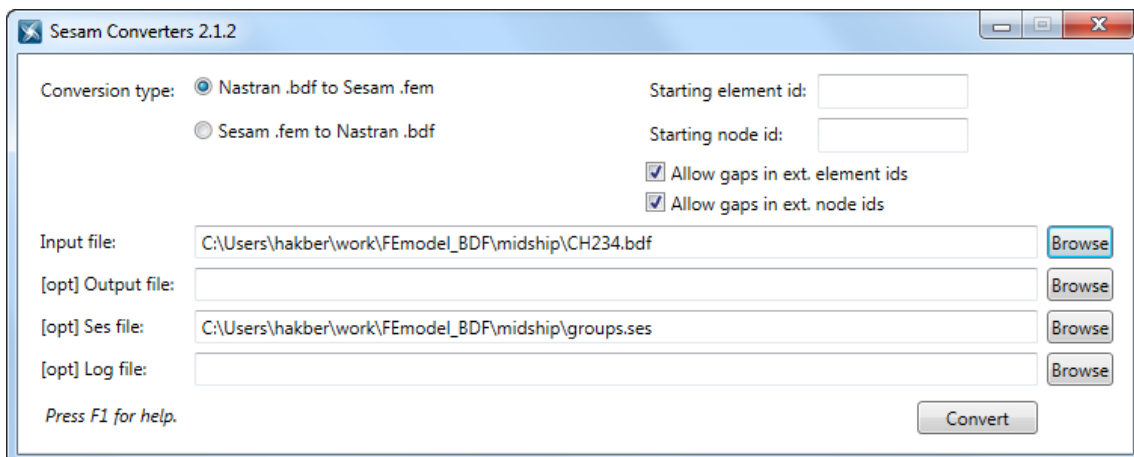
5.6 Load cases

To avoid any mixing of load case numbers later when using Nauticus Hull load cases and HGLA, it is useful to delete or exclude any load cases when exporting the Nastran model file.

5.7 Sesam Converters

Check the **Sesam Converters log file** for details and errors. The file is found in the workspace folder.

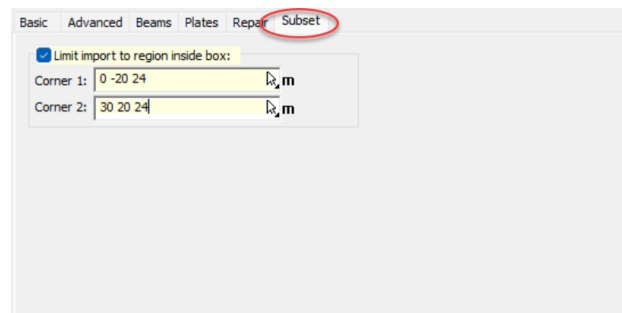
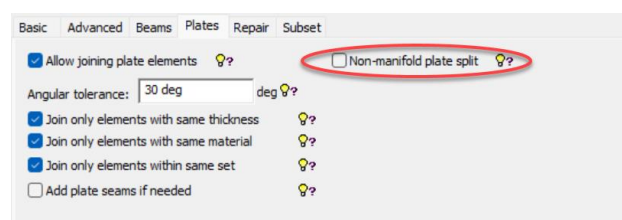
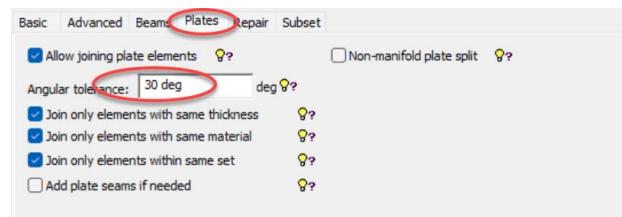
Sesam Converters can also be run **standalone**, enabling renumbering options that are not available from within GeniE:



If node and element ids are high (> 1 000 000) it may be beneficial to renumber to go from 1 and upwards without gaps. This can be done before export of the Nastran model or in Sesam Converters run standalone.

5.8 FEM import details

- During FEM import a small concept beam or plate/shell is created for each element, and then the concepts are **automatically joined** with adjacent concepts according to the default criteria.
- Note the default beam and plate joining tolerances and options (Beam and Plate tabs). The **angular tolerances** may be tightened if needed.
- If **groups/sets are not imported**, the **Non-manifold plate split** option should normally be checked so that the outer shell and inner side are split at all web frames and horizontal girders/stringers.
- Note the option to **limit the model import** to only what is inside a box.



About DNV

We are the independent expert in risk management and quality assurance. Driven by our purpose, to safeguard life, property and the environment, we empower our customers and their stakeholders with facts and reliable insights so that critical decisions can be made with confidence. As a trusted voice for many of the world's most successful organizations, we use our knowledge to advance safety and performance, set industry benchmarks, and inspire and invent solutions to tackle global transformations.

Digital Solutions

DNV is a world-leading provider of digital solutions and software applications with focus on the energy, maritime and healthcare markets. Our solutions are used worldwide to manage risk and performance for wind turbines, electric grids, pipelines, processing plants, offshore structures, ships, and more. Supported by our domain knowledge and Veracity assurance platform, we enable companies to digitize and manage business critical activities in a sustainable, cost-efficient, safe and secure way.