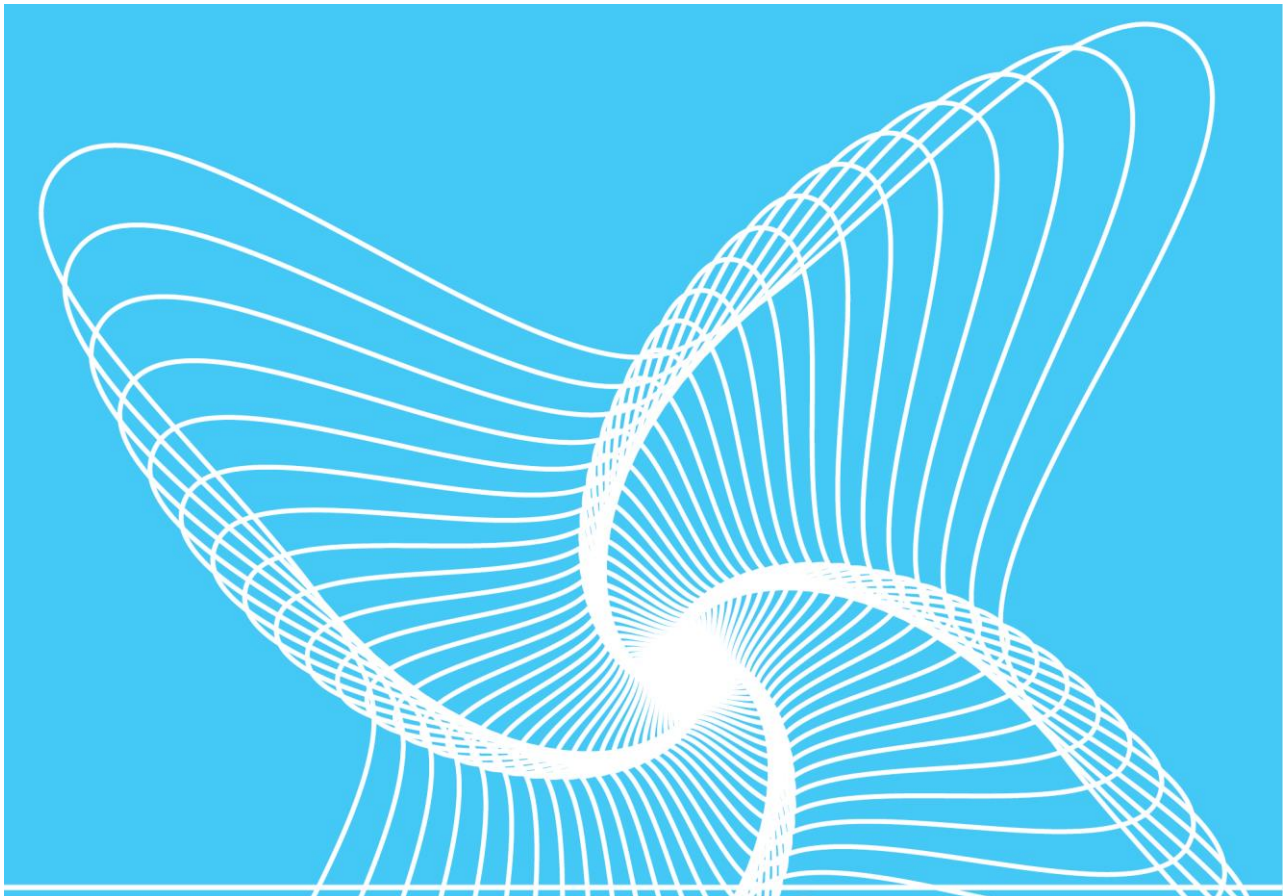DNV·GL

Sesam™

Guideline for Partial Meshing

Valid for GeniE version 7.7

Reference to part of this report which may lead to misinterpretation is not permissible.

No.	Date	Reason for Issue	Prepared by	Verified by	Approved by
0	6 June 2014	First issue	Nikolay Qviller	Håkon Berg	Ole Jan Nekstad
1	8 January 2015	Updated limitations	Jan Land	Nikolay Qviller	Ole Jan Nekstad
2	9 January 2015	Added note on "Keep mesh"	Nikolay Qviller	Atle Westvang	Ole Jan Nekstad
3	25 June 2015	Updated limitations	Nikolay Qviller		
4	20 December 2017	Updated graphics and text	Nikolay Qviller		

20 December 2017

Prepared by DNV GL - Software

Table of contents

1   INTRODUCTION

By "partial meshing" we mean "partial update of mesh" as opposed to "regeneration of mesh". GeniE has long supported "conditional remeshing" for property updates (where the program decides whether to update loads and/or remesh). With partial meshing we extend this feature to cover more properties and also geometrical model updates.

The main advantages of the partial mesher are:
- Speed-up since only geometrically modified parts of the model are remeshed
- Preserved node and element numbering for untouched parts
- More user control of the mesh at selected parts

Partial meshing consists of:
- A mapping between the concept model and the mesh which tracks geometrical modifications and property updates
- A new high-level mesher which uses the above mapping and a set of rules described below for determining precisely which parts of the model should be remeshed

Partial meshing is always activated, but the user may choose between "Always Regenerate" and "Conditional Regenerate", the last option leaving to GeniE to decide which parts of the model need remesh.
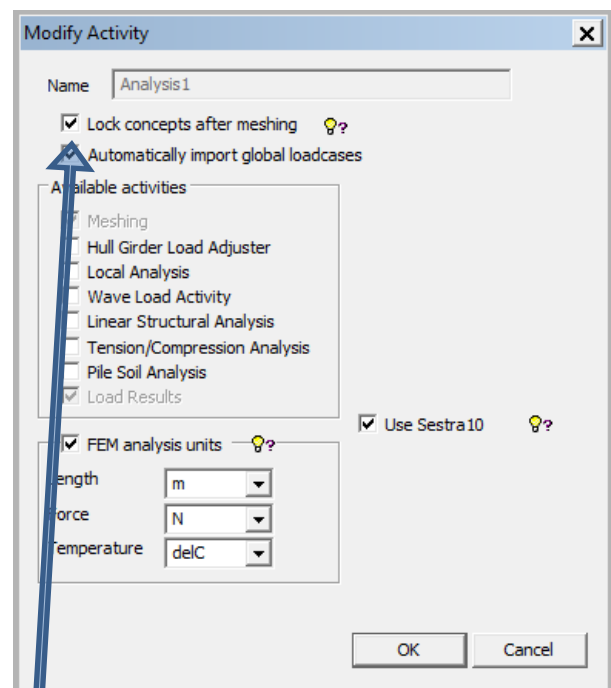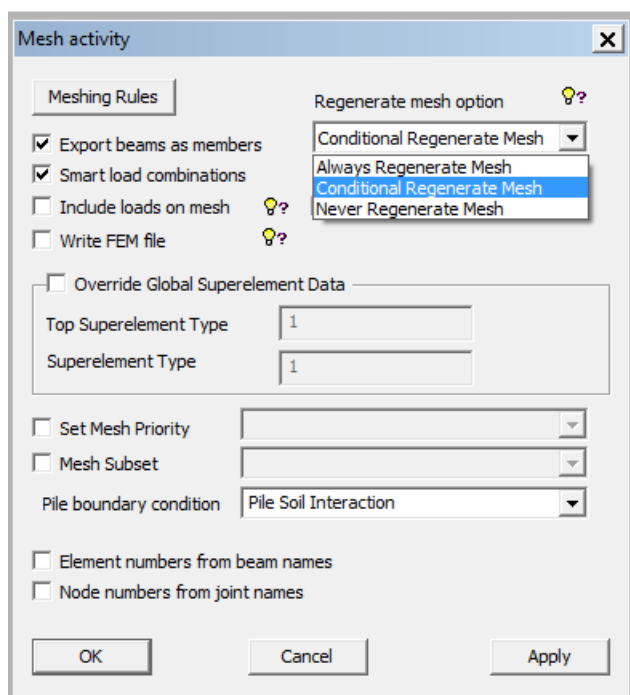
2 PARTIAL MESHING

## 2.1 Rules for automated remeshing

In the following, by "geometrically modified" we mean a modification that will change the finite element geometry, i.e.,

- Splitting of existing topology (e.g. inserting a feature edge splitting a plate)
- Removing or inserting concepts (e.g. inserting a new beam or plate)
- Merging of topologies (e.g. joining two plates and simplifying topology)
- Changing mesh option or mesh property of concepts (e.g. increasing the mesh density, changing face mesher or edge mesher)

With that definition at hand, the following rules are used to determine which topological entities need to be remeshed:

- A vertex (topological point) is meshed if it was previously unmeshed, and remeshed if all edges incoming to that vertex are triggered for meshing/remeshing
- An edge (topological line) is meshed if it was previously unmeshed, or is remeshed following these rules:
  - If it has been joined with an unmeshed entity and has no other modification, it is only remeshed if required for a consistent mesh
  - If it has any other geometric modification, it is always remeshed
  - If no modification has been done, the following check is carried out: Is the edge part of a previously meshed and unchanged beam?
    - If yes: Don't remesh
    - If no, remesh if and only if all faces sharing that edge need remesh
- A face (topological surface) is meshed if it is previously unmeshed, or is remeshed following these rules:
  - If it has been joined with an unmeshed entity and has no other modification, it is only remeshed if it has an edge without mesh
  - If it has any other geometric modification, it is remeshed
  - If not, it is remeshed if any part of its boundary has been geometrically modified

"Conditional remesh" is default and only the child activities required at a given time are checked in the Activity Monitor.
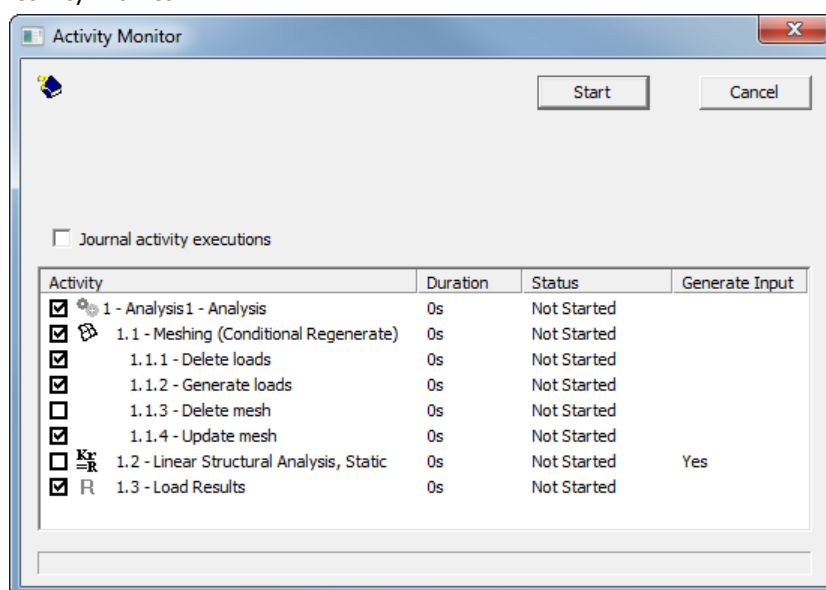
2.2 Locking of meshed concepts

The default setting in partial meshing mode is that concepts are locked after they have been meshed, thereby preventing propagation of mesh densities into neighbouring concepts during mesh update. To be more precise, if two neighbouring plates have been meshed and the mesh density is refined on one of them, this will NOT propagate to the other plate. Use "Force remesh" or "Force remesh including neighbours" (see chapter 3) to have the mesh changes propagate outside the modified part of the model, i.e. to get a mesh transition zone outside the modified area.

IMPORTANT: The interior mesh of a plate is determined by the mesh of its boundary. Thus, if the entire boundary of a plate is locked in the above sense, a change to its mesh density will NOT result in a visible change in the interior net. See Section "Forced remesh" for more on this.

The default behaviour can be switched OFF in the "Modify linear analysis" dialog.

REMARK: It may be that the user wishes to keep the default behavior ON and still refine the density on a plate whose entire boundary is locked. For this purpose, it is recommended to use the Guiding Geometry -> Curves on Surfaces -> Plate/Shell Edges' Offsets feature to offset the boundary on the plate, followed by creating feature edges on the offset boundary. If the plate consists of several faces, it must be exploded into simpler parts before carrying out this step individually for each face.

NOTE: A beam is not locked by surrounding plates. Changing the mesh density of a beam will therefore affect the mesh of neighbouring plates.

2.3 Automatic triggering of remesh/mesh update of a region

The following actions will trigger a partial update of the mesh of the affected concepts, e.g. a remesh or an update of properties:
- Deleting concepts (triggers removal of mesh)
- Adding concepts (triggers meshing of added concepts and affected neighbours)
- Joining concepts (triggers remesh and/or update of concept information on FEM file)
- Splitting concepts (triggers remesh)
- Changing mesh subset (triggers update of mesh and meshing of previously unmeshed concepts)
- Inserting feature concepts, support points, support curves, point masses, joints etc. (triggers local remesh if actions have modified underlying geometry)
- Geometry changes, e.g., moving a beam end point (triggers remesh)
- Adding mesh properties to a concept or changing existing mesh properties (triggers remesh)
- Flipping a beam or plate normal (triggers property update)
- Changing plate or beam type (triggers remesh)
- Adding/changing sections, thicknesses, hinges, eccentricities (triggers property update)
- Adding/changing corrosion addition, effective flange, net scantlings (triggers property update)

3 FORCE REMESH OF A PART OF THE MODEL

The user can force remesh of primary concepts (beams and plates) which are otherwise untouched by right clicking and selecting "Force remesh" or "Force remesh including neighbours". This will trigger remesh in the current analysis. Reset this by using "Do not force remesh". Note that the latter will not override automatic triggering of remesh.
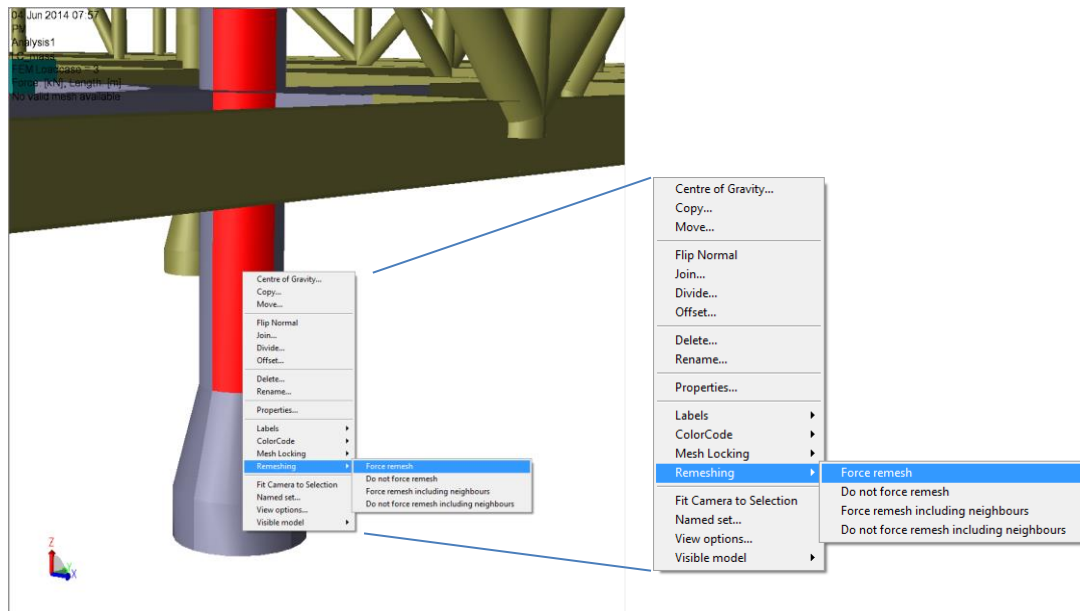
The browser shows which concepts are triggered for remeshing under "Needs remesh". These can also be colour coded for easy graphic detection. Please note that the browser may require a manual update after model changes.

- For performance reasons, the algorithm used to update the status in the browser is "superficial" and may report that a concept needs remesh even though it will not necessarily be remeshed. Example: Adding a beam on the boundary of a meshed plate will mark the plate as needing remesh, even though a full remesh is not necessary
- The browser will not report "Needs remesh" for changes which do not affect the finite element geometry, such as changing the section of a beam


In the example above, the geometry has been changed for Bm1 -> Bm6 after the first meshing. The beams are now marked "Needs remesh" and will be remeshed next time the mesh is generated.

NOTE: There are a few scenarios where a manual use of "Force remesh" may be necessary to achieve a consistent mesh. For instance, if the user has modelled two partially overlapping, disconnected beams and meshed them, then removes the "Disconnected" property from the inner beam, it is necessary to use "Force remesh" on the outer beam to achieve a consistent mesh.

4 FORCED REMESH OF WHOLE MODEL

As previously mentioned, "Conditional remesh" is the default choice for all mesh activities, meaning only affected parts of the model are remeshed or updated at each activity execution.
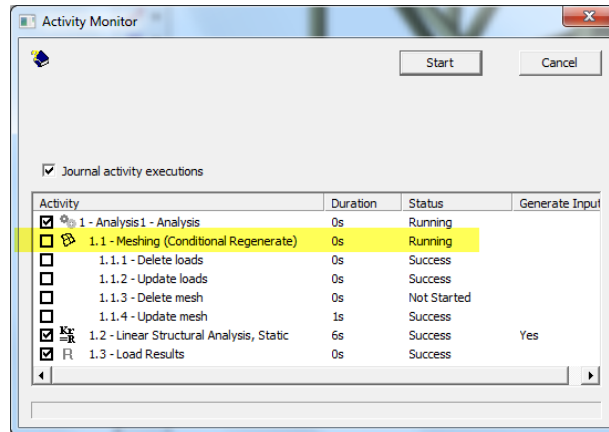
The user can switch to "Always regenerate" to use the partial mesher for a full regeneration of the mesh.
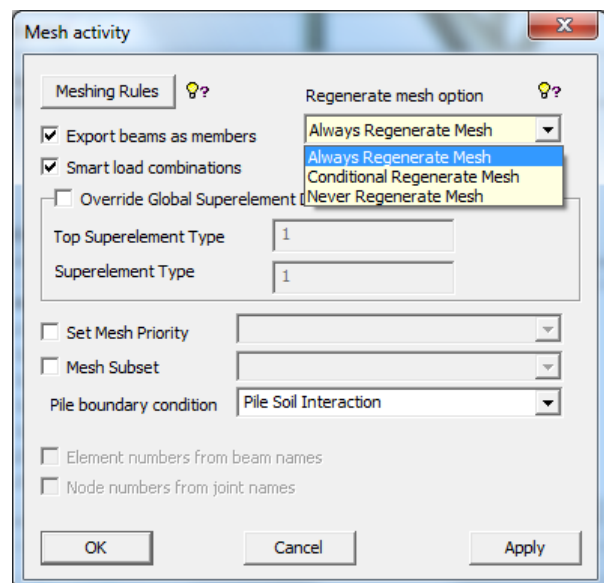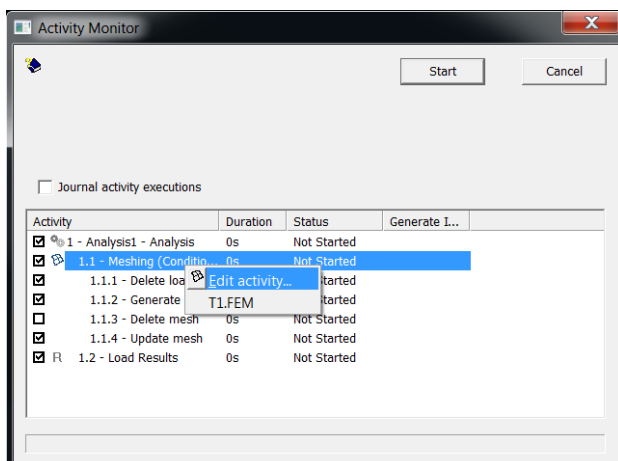
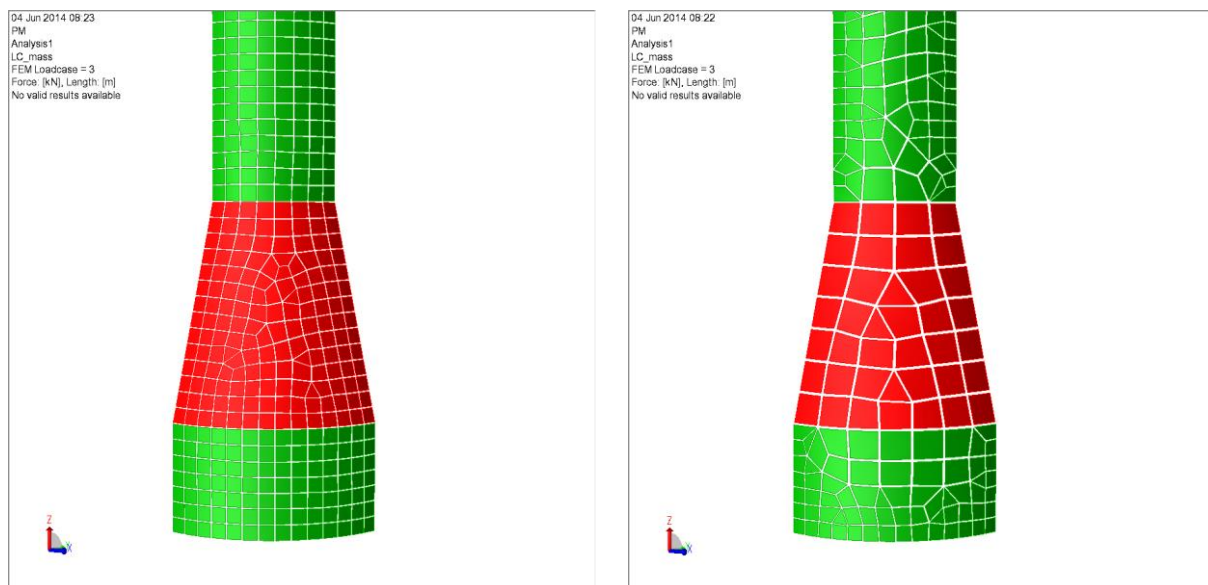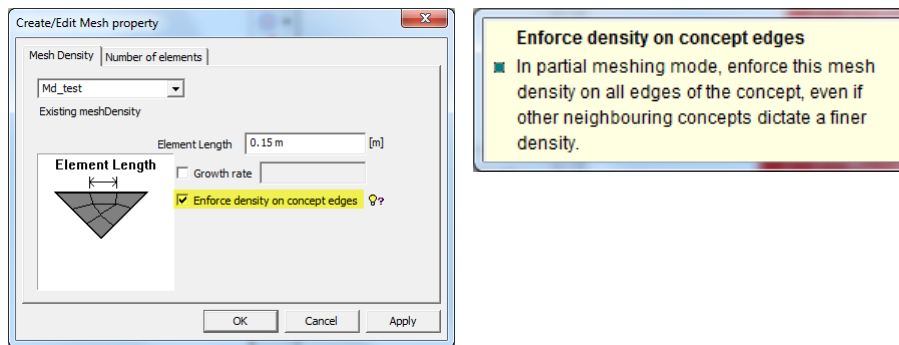A full remesh of the model will be triggered if any of the following applies:
- User switches from first to second order elements or the other way around.
- User turns "Use drilling elements" ON or OFF when it was previously OFF or ON
- User changes face mesher
- User changes edge mesher
- User deletes all concepts
- User changes idealization options
- User changes meshing rules (quality, "Allow triangles" etc.)

5 ENFORCING A COARSE DENSITY ON BOUNDARY SHARED BY TWO PLATES

In partial meshing, mesh densities can be enforced on a common boundary between plates triggered for remesh. This will allow the user to let a coarse density override a fine density on the common edge between two plates, for optimal control of the propagation zone. Without turning this option on, mesh densities will never be overridden if they are coarser than the global default density or a density specified for an attached concept.

Enforce density on concept edges
■ In partial meshing mode, enforce this mesh density on all edges of the concept, even if other neighbouring concepts dictate a finer density.

In the example above finer mesh density has been applied to the upper and lower surface. In the left picture the "Enforce density on concept edges" has been deactivated (this is also the default setting).

NOTE: If a plate is locked by surrounding meshed plates, switching on "Enforce density on concept edges" is not enough to update the mesh of the plate and its surroundings. The user must specify which neighbouring concepts to trigger for remesh using "Force remesh" or use "Force remesh including neighbours" on the plate in question to trigger remesh of all its neighbouring plates.

6 THE GNX WORKSPACE FILE

6.1 Purpose

The GNX file is designed for exchange of concept models and analysis mesh data. It is the preferred format for exchange to a newer GeniE version due to portability reasons, efficiency and enhanced features. The GNX can be specified to include "Mesh on XML"; with this feature, the mapping between the concept model and the mesh is preserved for continued work upon import. This is also the system default setting.

> - Include mesh on XML means that the mesh will be kept for the individual analyses and will be restored during import.
> - Please note that only analyses that are completely meshed, i.e. no mesh update needed in activity monitor, can be exported. Due to current limitations in partial meshing, analyses with piles, inner beams and spring connections in their subset cannot be exported.

6.2 Format

A GNX file is a compressed archive (zip format), containing model data (on XML), geometry and mesh information. The XML format used within the GNX is the same as the one for XML concept model files and therefore has the same capabilities and limitations.

6.3 Export

During export of a workspace to a GNX file, only analyses which have been fully meshed (i.e. no update needed in the activity monitor) can be exported if the user checks "Include mesh on XML". This will ensure that the mapping between a concept and its finite element mesh is kept for later use of the data, for example when importing to a new workspace.

6.4 Import

When a GNX file is imported into an existing workspace, models are merged together. In case of name conflicts, new names are generated for imported entities.
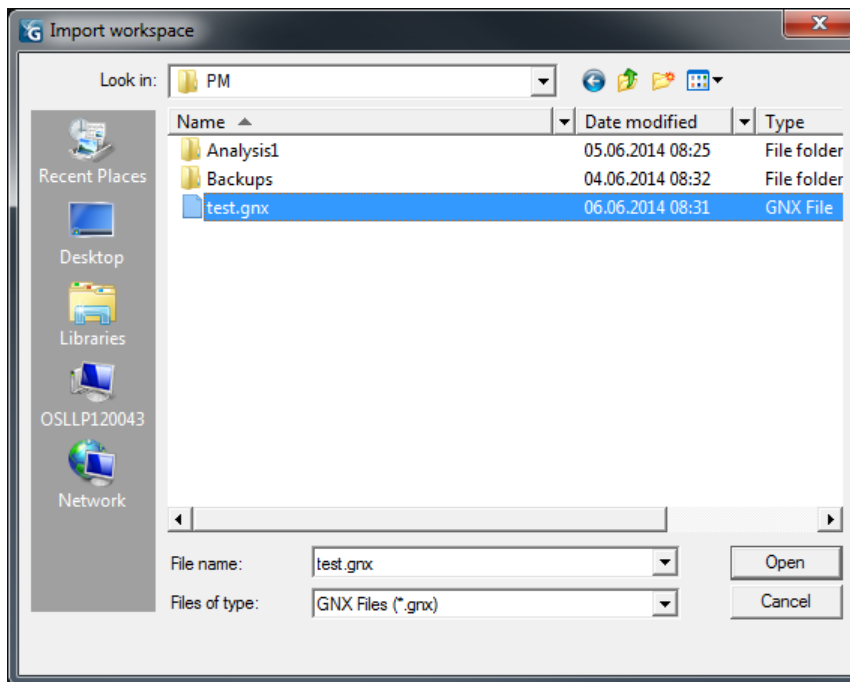
It is **not** recommended to merge analyses by importing multiple GNX files with possibly overlapping element numberings. Instead you should use XML import and perform a full re-mesh. The reason is that a first import of a GNX file will contain a mesh and subsequent GNX imports will also contain mesh that may conflict with the previous mesh.

7 KEEP MESH

"Keep mesh" is a feature enabled in partial meshing mode, allowing the user to:
- Select an active mesh subset of the model, mesh it and tweak the mesh until satisfied
- Store this mesh and switch to another "active" mesh subset
- Mesh and tweak this second active subset, seeing only this subset in model and mesh view (using Alt+S to reduce view to current subset), then store it along with the first mesh subset
- Continue this way until satisfied with several mesh subsets, all of which have been stored
- Ask for a full mesh of the entire model, but keeping the mesh of each of the individual subset already meshed.

It is **highly recommended** to use this on physically disjoint subsets (i.e., subsets which do not share any common topology except common plate boundaries).

Warning: "Keep mesh" should be used with caution when "Remove internal vertices" is enabled in meshing rules. This is because some vertices may be considered internal when two neighbouring structures are meshed together, but not considered internal when only on of them is meshed. Such situations could lead to non-consistent meshes.

To use "Keep mesh", enable partial meshing and subset meshing. When switching to a different subset, the user will be asked if mesh is to be kept. If user selects "yes", the mesh will remain along with the mesh of current subset.

When user switches off subset meshing and answers yes to keep mesh, all previously stored subset meshes will remain. The next partial meshing will only mesh unmeshed or modified parts of the model.

8 CURRENT LIMITATIONS IN PARTIAL MESHING

The following limitations apply to partial meshing in GeniE version 7.7:

- Exporting an analysis to GNX with mesh on XML requires the analysis to be fully executed. If a global loadcase is used, and user adds an analysis, existing analysis referring to the global loadcase will be marked for load and mesh update.
- Partial regeneration of loads has not been optimized for partial meshing, thus we recommend applying loads to a finalized mesh.
- If a global load case is used and the user adds an analysis, existing analyses referring to the global load case will be marked for load and mesh update.
- Modifying a corrosion addition property assigned to a plate will not trigger "Update mesh". If this is the only change required by the user before re-running an analysis, we recommend replacing the corrosion addition with another one; this action will trigger "Update mesh".
- Turning the option "Include unused properties" ON or OFF does not trigger partial update of mesh.
- Changing the intensity of the moment part of a point load applied to a beam will not trigger removal of the footprint node. To achieve this use "Force remesh".

For the major limitations, and some of the minor limitations, warnings are produced in the Activity Monitor. It is therefore recommended to use the Activity Monitor to run analyses to see warnings if any (instead of using scripting).

ABOUT DNV GL

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

SOFTWARE

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