

SESAM TUTORIAL

GeniE

Mesh Refinement Box

Valid from program version 8.2





Sesam Tutorial

GeniE – Mesh Refinement Box

Date: October 2021

Valid from GeniE version 8.2

Prepared by: Digital Solutions at DNV

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

E-mail sales: digital@dnv.com

© DNV AS. All rights reserved

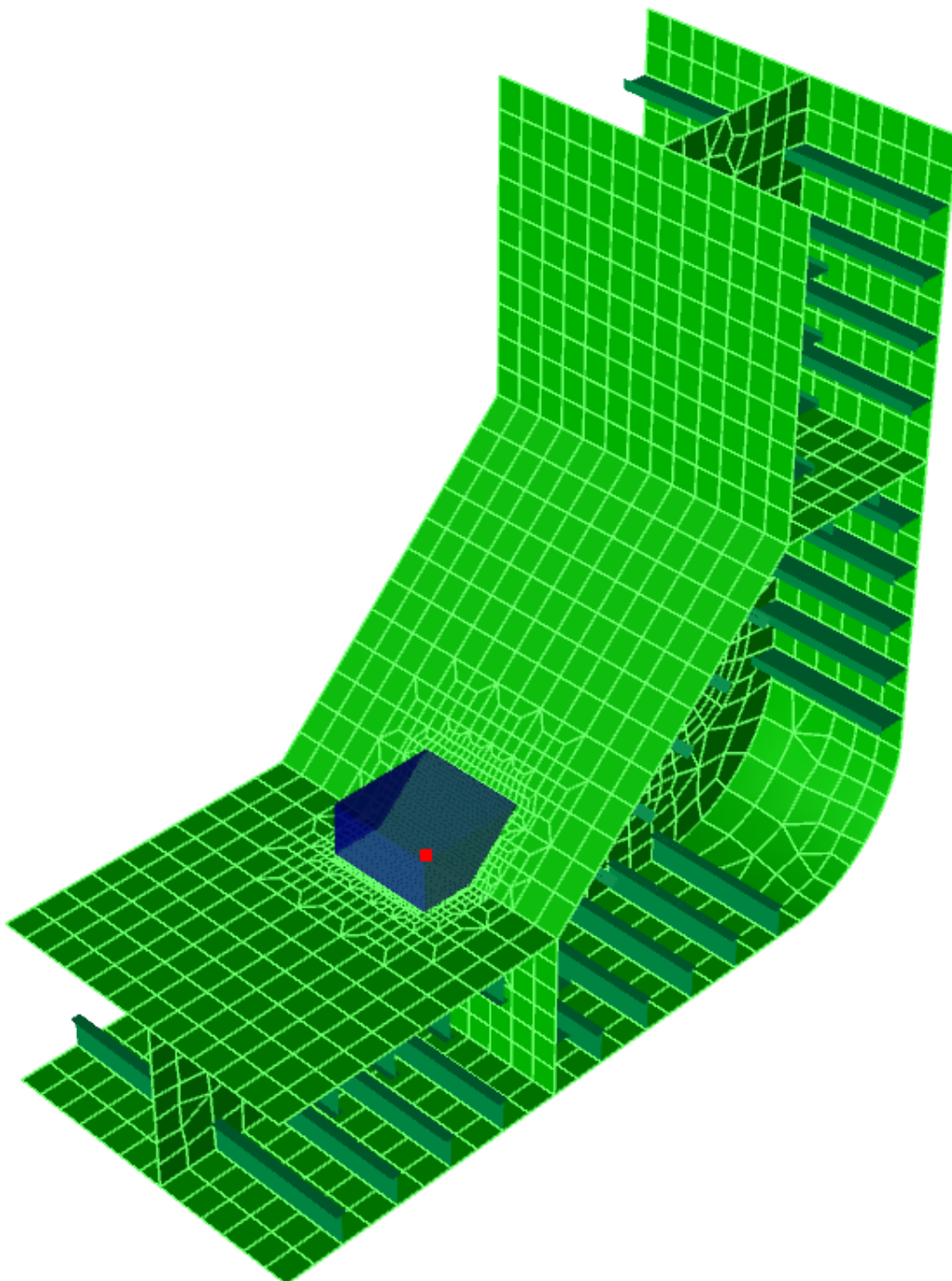
This publication or parts thereof may not be reproduced or transmitted in any form or by any means, including copying or recording, without the prior written consent of DNV AS.

TABLE OF CONTENTS

1. Introduction	Page 4
2. Create a New Workspace and Import a Model	Page 5
3. Create Hole in Plate and Use <i>Mesh Option for Hole</i>	Page 8
4. Refinement Within a Defined Area Using the <i>Refine Box</i> Feature	Page 11

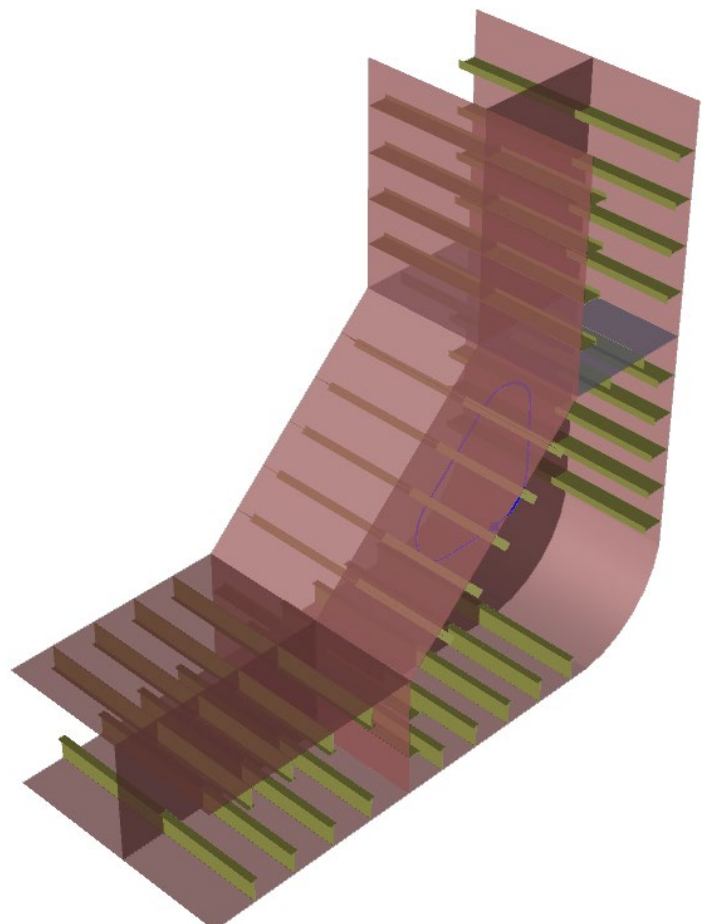
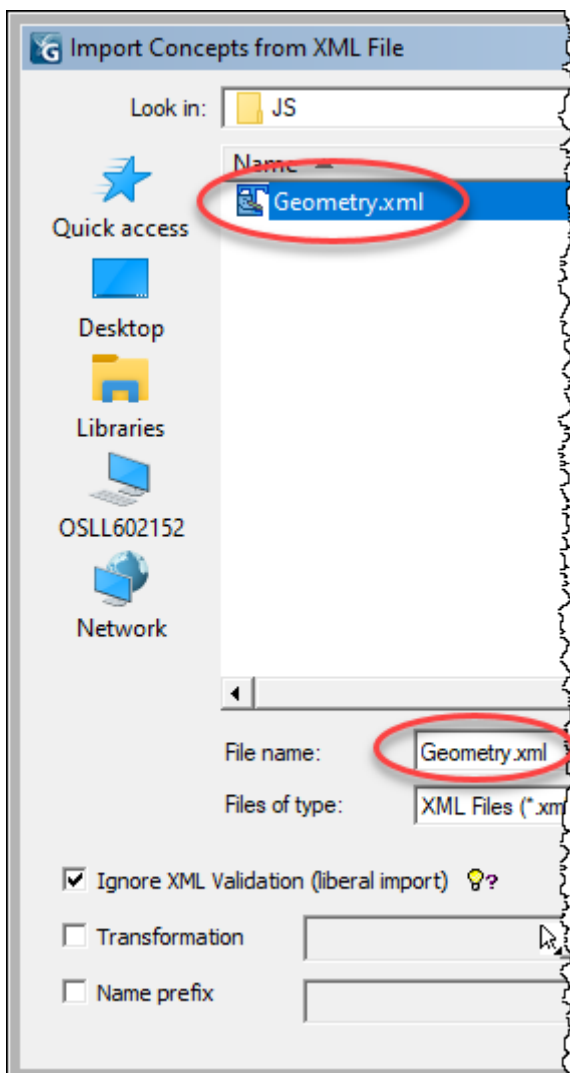
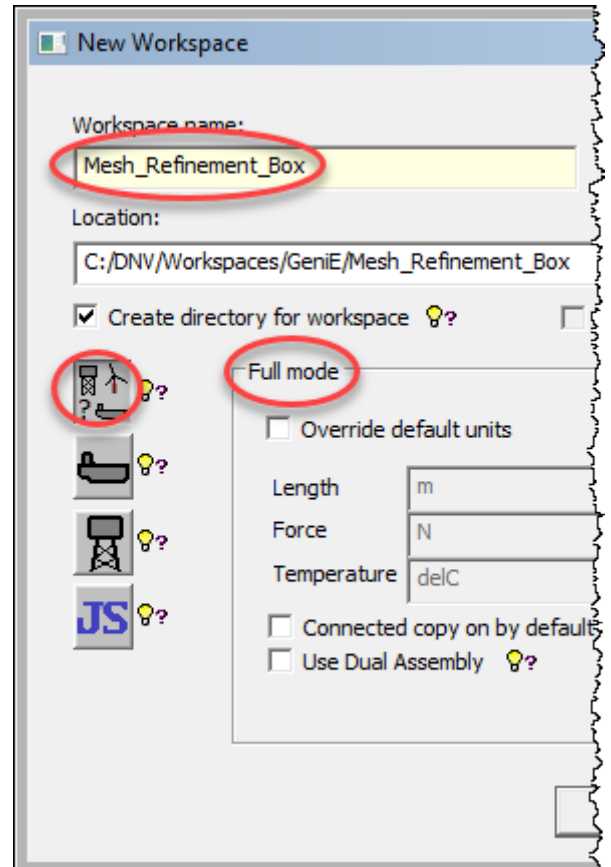
1 INTRODUCTION

- This tutorial explains how to refine the automatically created mesh by two different features:
 1. Refinement around a hole by using the *Mesh Option for Hole* feature
 2. Refinement within a defined area by using the *Refine Box* feature
- A model of a simplified ship section is imported for the purpose of this tutorial.
- Note that there are more features for mesh editing than covered by this tutorial. Go to other tutorials to learn about such mesh editing features.

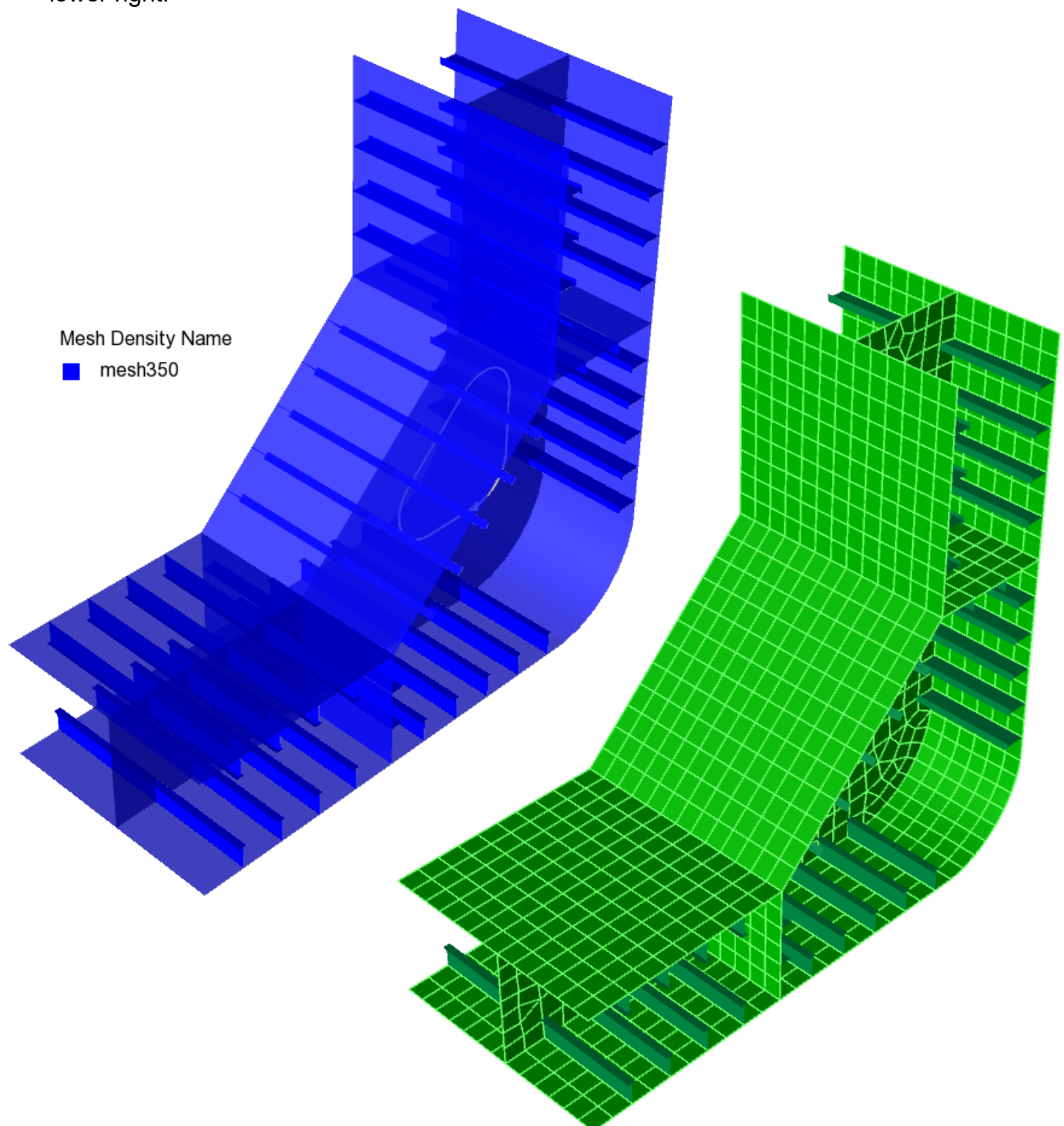


2 CREATE A NEW WORKSPACE AND IMPORT A MODEL

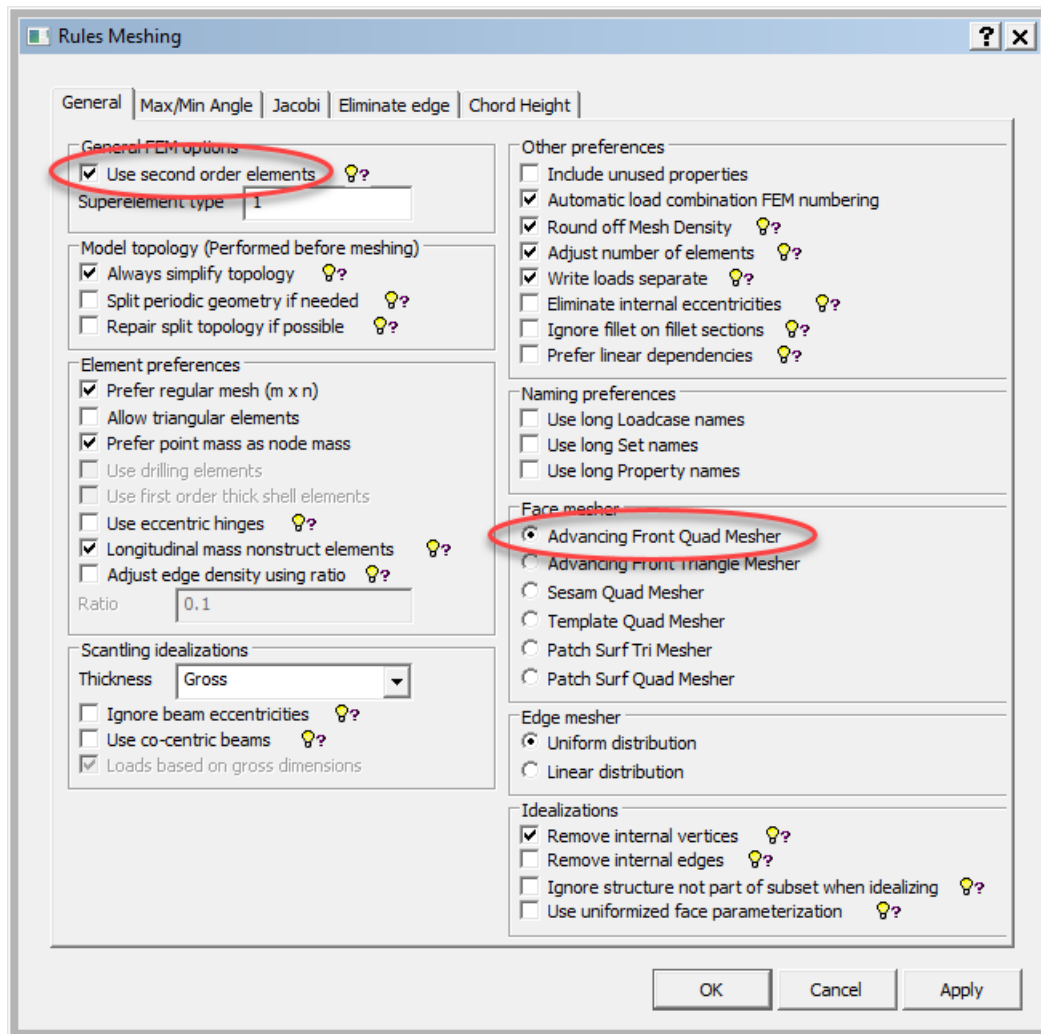
- Start GeniE and open a new workspace.
 - Give a workspace name, for example Mesh_Refinement_Box.
 - Press the *Full mode* button to open for curved geometry modelling.
 - Accept default units m and N and click OK.
- Use *File | Import | XML Concept Model* to import the file Geometry.xml. The file is found as part of the installation, typically at:
C:\Program Files\DNV\GeniE VX.Y-ZZ\Help\Tutorials\TutorialsAdvancedModelling\A14_GeniE_Mesh_Refinement_Box\JS
- The model shown to the lower right appears.



- The imported model includes mesh properties (see folder *Properties* | *Mesh* in the browser) but no mesh properties have been assigned to the model.
- Select the whole model (optionally by selecting the set named `concept_all` found in the folder *Utilities* | *Sets* | *Regular Sets* in the browser) and assign the mesh property named `mesh_350` to the model. Mesh properties are colour coded below.
- Use Alt+M to open the *Activity Monitor* and click *Start* to create the FE mesh shown to the lower right.

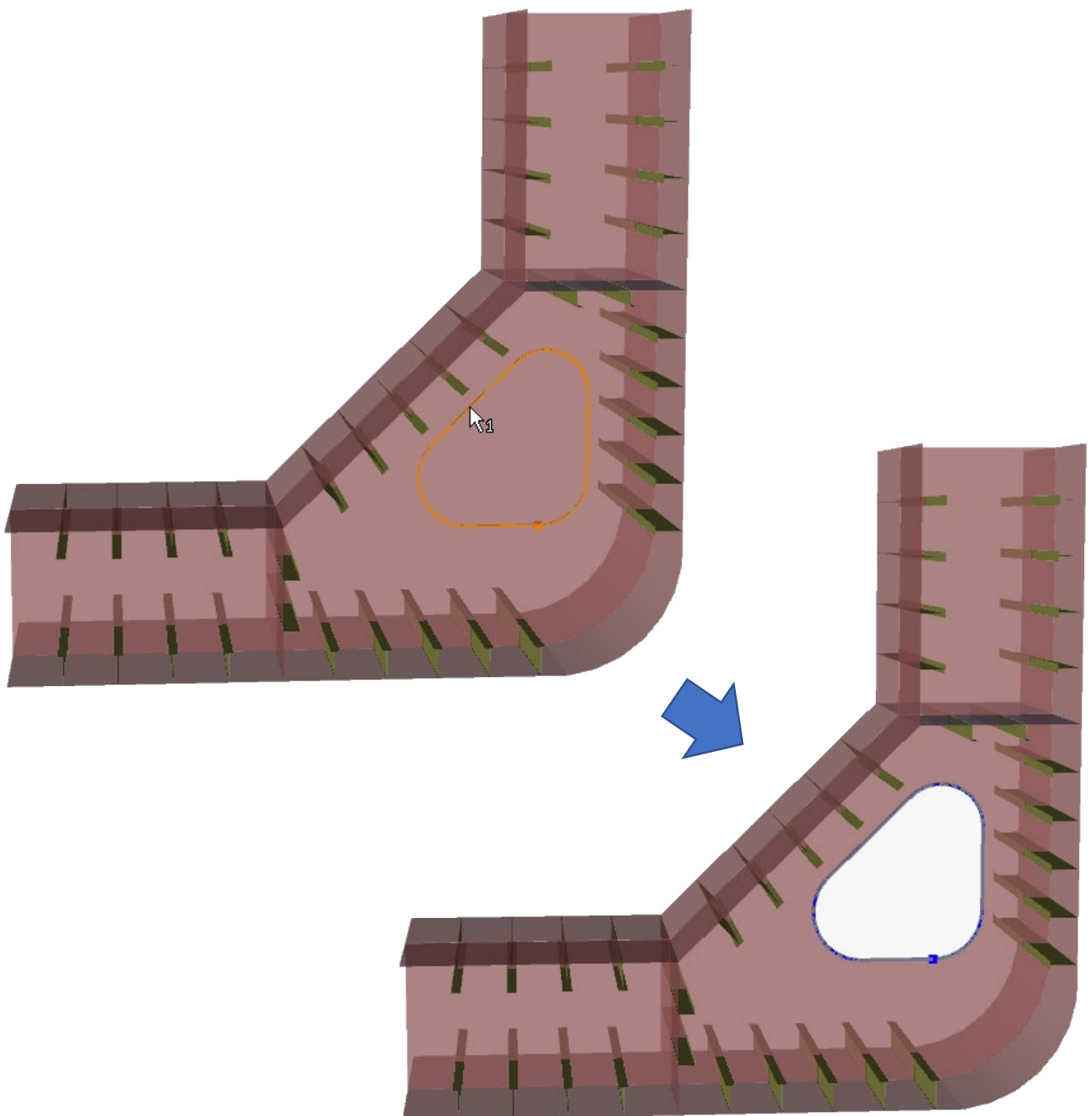


- Note that the mesh has been created using the *Advancing Front Quad Mesher* and that second order (eight-node quadrilateral and six-node triangular) elements are used. This is seen by *Edit | Rules | Meshing Rules*.



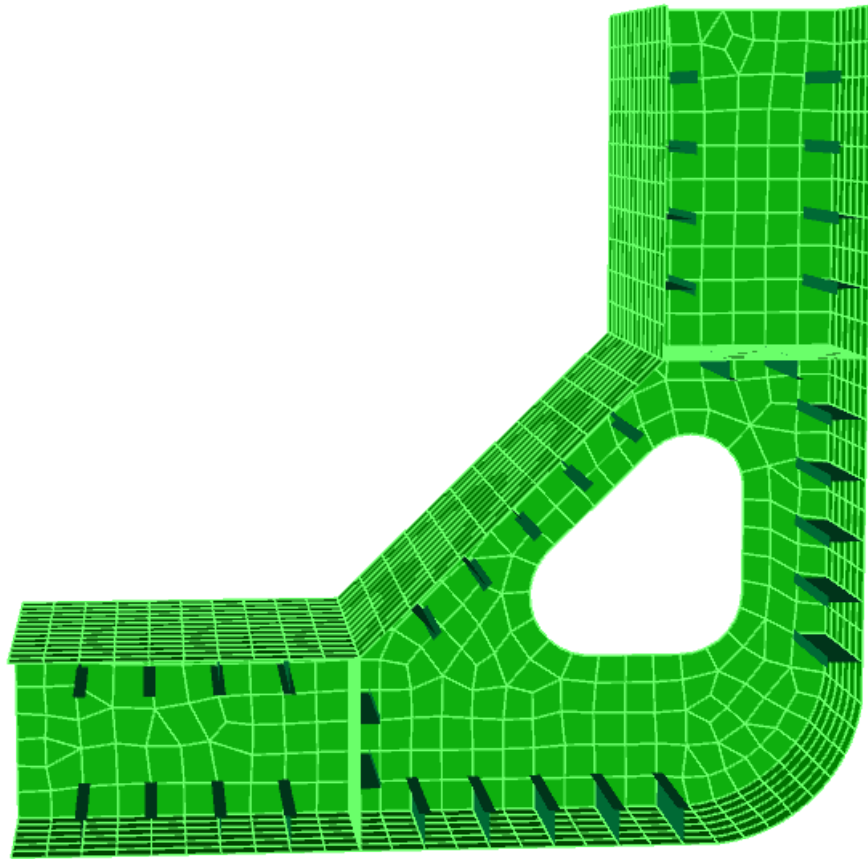
3 CREATE HOLE IN PLATE AND USE *MESH OPTION FOR HOLE*

- Use *Structure | Features | Hole from Guiding Curve* and click the guiding curve to create a hole in the plate as shown below.
- Note that this creates a hole concept that differs from a regular hole in the plate by being an object that, when deleted, the hole is no longer there.
 - Find the hole concept in the folder *Structure | Holes* in the browser.
 - Just as a test, try deleting it and see the hole is no longer there.
 - Use Ctrl+Z to undo the deletion, the hole reappears..

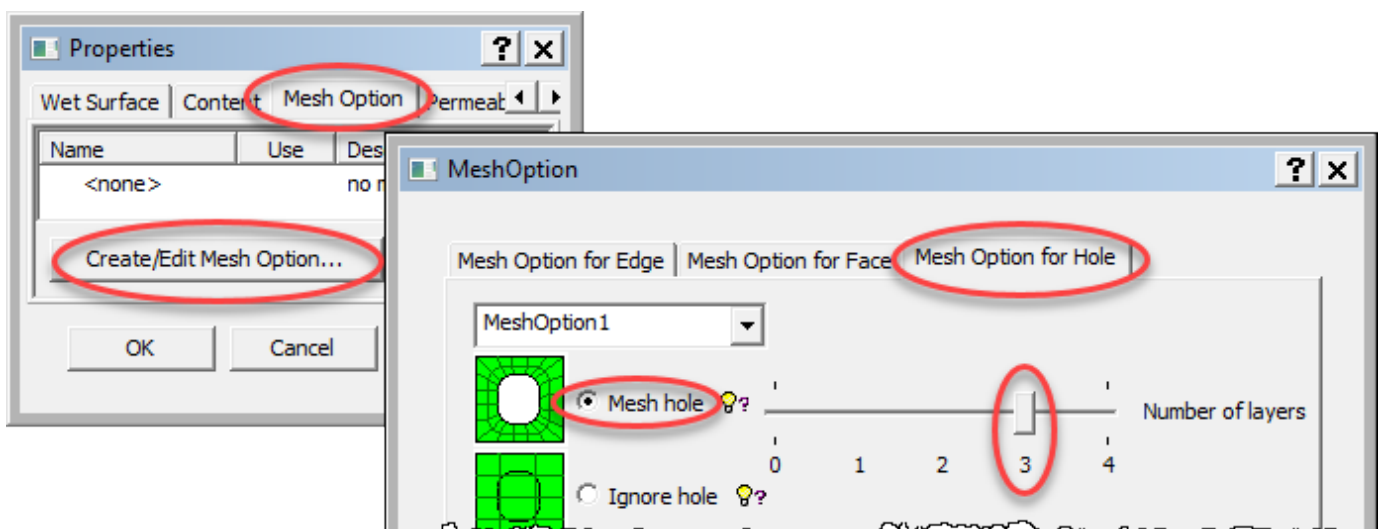


3 CREATE HOLE IN PLATE AND USE *MESH OPTION* FOR HOLE

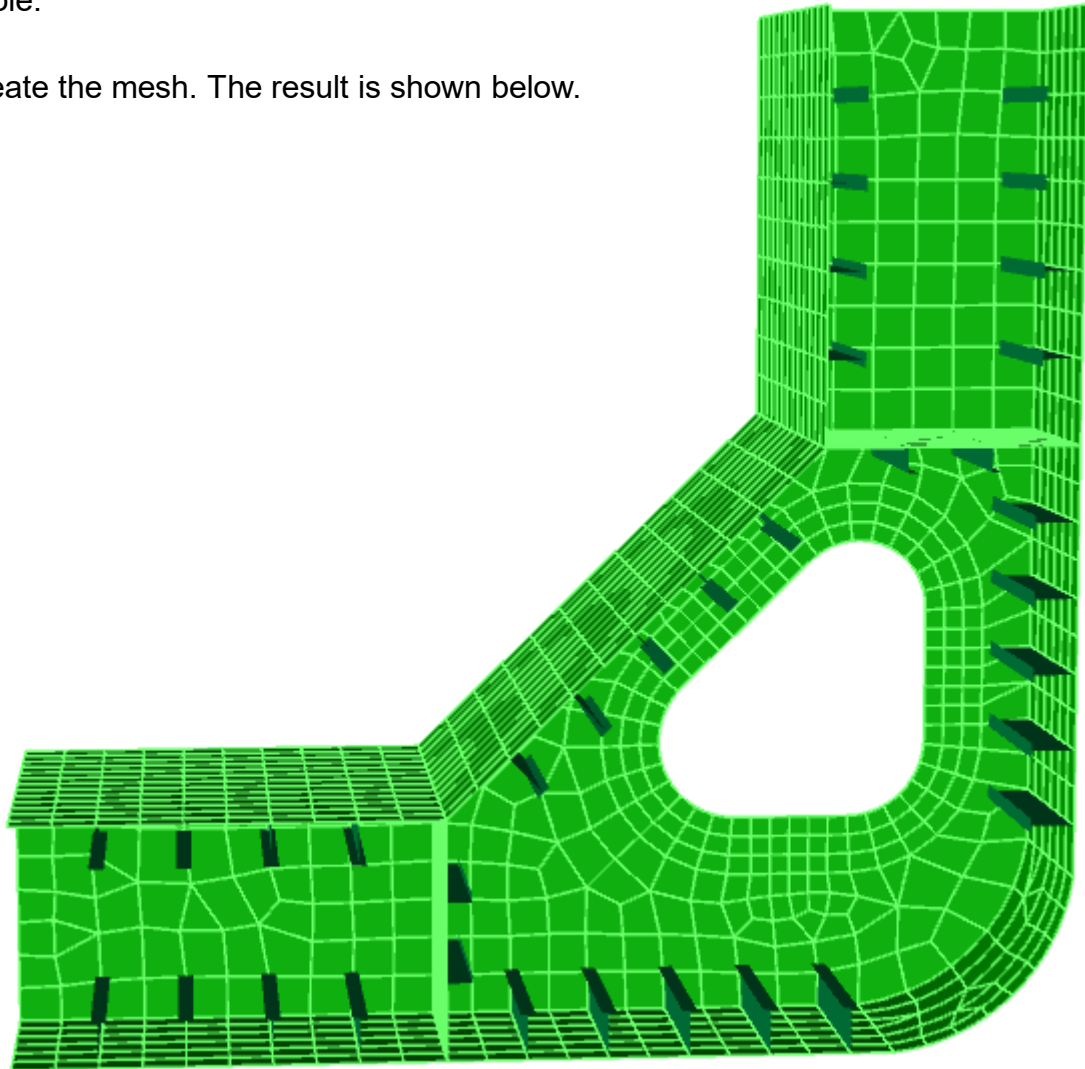
- Recreate the mesh. The result is shown below.



- The feature for automatic mesh generation will in most cases create satisfactory mesh around holes. However, in certain cases finer mesh around holes are required.
- Use *Edit | Properties* to open the *Properties* dialog. Go to the *Mesh Option* tab and click *Create/Edit Mesh Option* to open the *Mesh Option* dialog.
- In the *Mesh Option for Hole* tab, select *Mesh hole* with *Number of layers* set to 3.



- Select the hole in the *Structure | Holes* folder and assign the newly created mesh option named *MeshOption1*, found in the *Properties | Mesh Options* folder, to it.
- Also assign the mesh property named *mesh200*, found in the *Properties | Mesh* folder, to the hole.
- Recreate the mesh. The result is shown below.

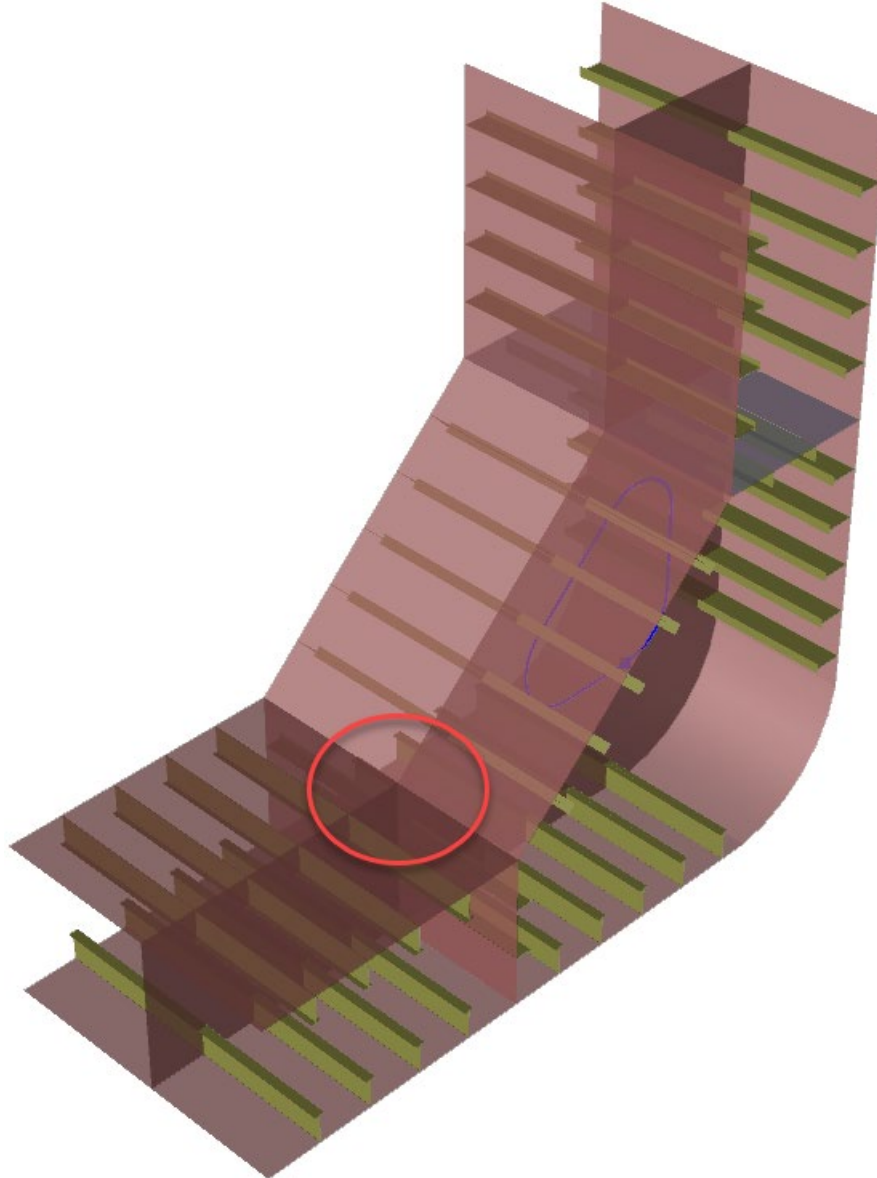


- There are other methods for creating holes and refining the mesh around it.
 - Holes can be created by deleting the 'plate inside the hole'.
 - Guide curves may be created around holes by *Guiding Geometry | Curves on Surfaces | Plate/Shell Edges' Offsets on the Surface* followed by creating feature edges based on these guide curves. Mesh properties may then be assigned to the feature edges.
 - Manual mesh editing and refinement may be done using the tools for manual mesh editing found in the *Edit Mesh* toolbar shown below. There is a tutorial dedicated to such mesh editing.
 - Such mesh refinement may also be done to the mesh above, e.g. using *Refine Grid Including The Boundary (2)*.



4 REFINEMENT WITHIN A DEFINED AREA USING THE *REFINE BOX* FEATURE

- Now we want to refine the mesh in the encircled area shown below. The reason for this might be that a detailed mesh is required for a fatigue analysis.



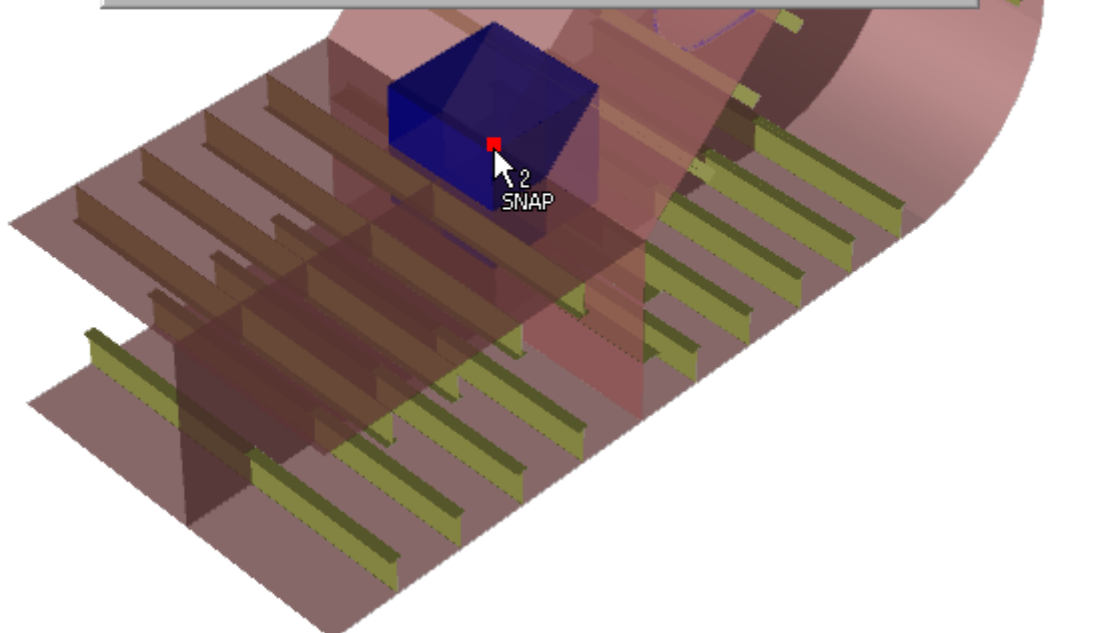
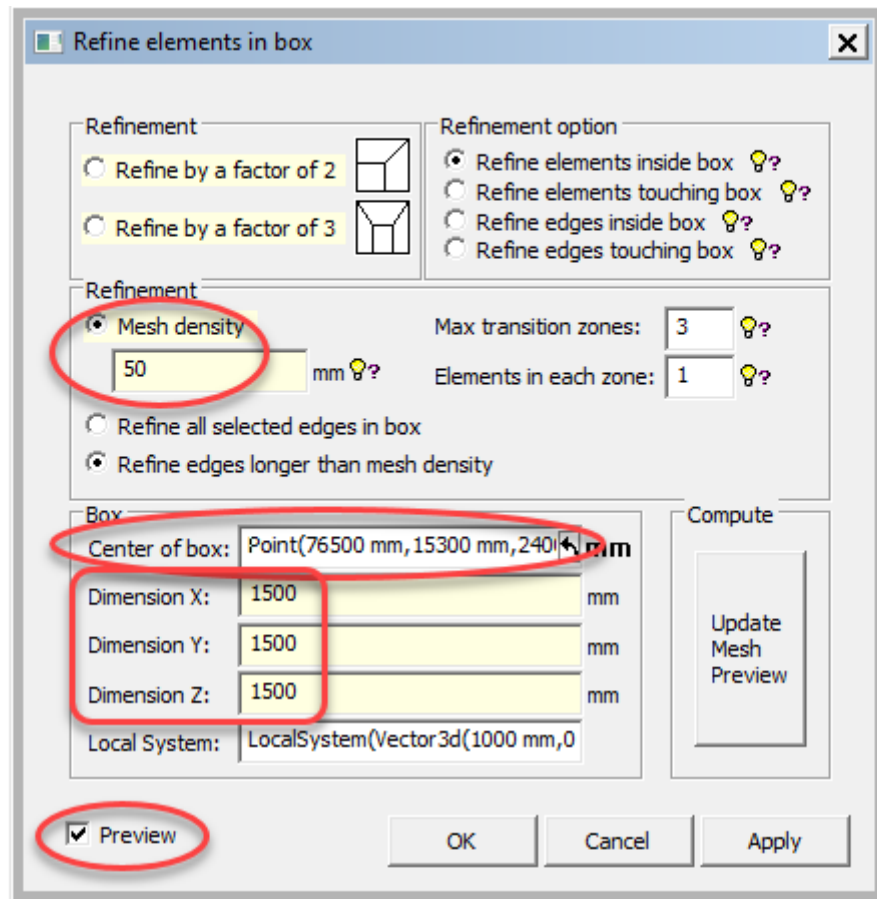
- Rather than modifying the concept (geometry) model in the area of interest, creating new sets and applying several different mesh properties to these sets, a feature named *Refine Box* can be used.
 - By this tool a box (volume) is defined with certain mesh refinement settings.
 - The refinement setting can be:
 - Refinement factor of 2 or 3 to the existing mesh
 - Refinement by setting maximum element size
 - A proper transition from the refined mesh within the box to the original coarser mesh outside the box is created as part of the process.

- Use *Mesh & Analysis | Mesh Editing | Refine | Refine Box (Dialog)* or click the *Refine Box (Dialog)* button to open the *Refine elements in box* dialog shown below.

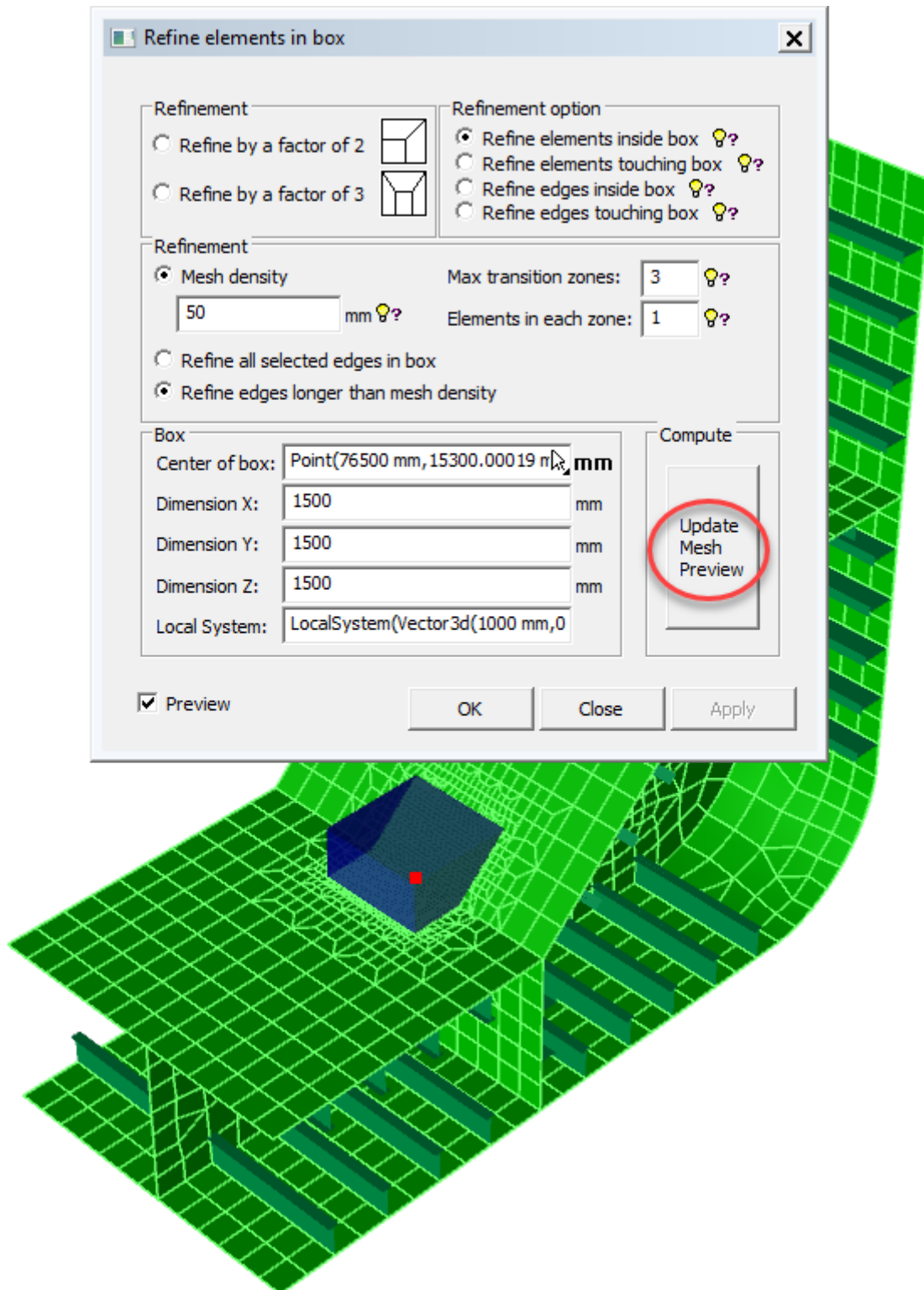


- Click *OK* to the warning about script commands.

- Check *Preview* to see the dimensions and position of the refinement box.

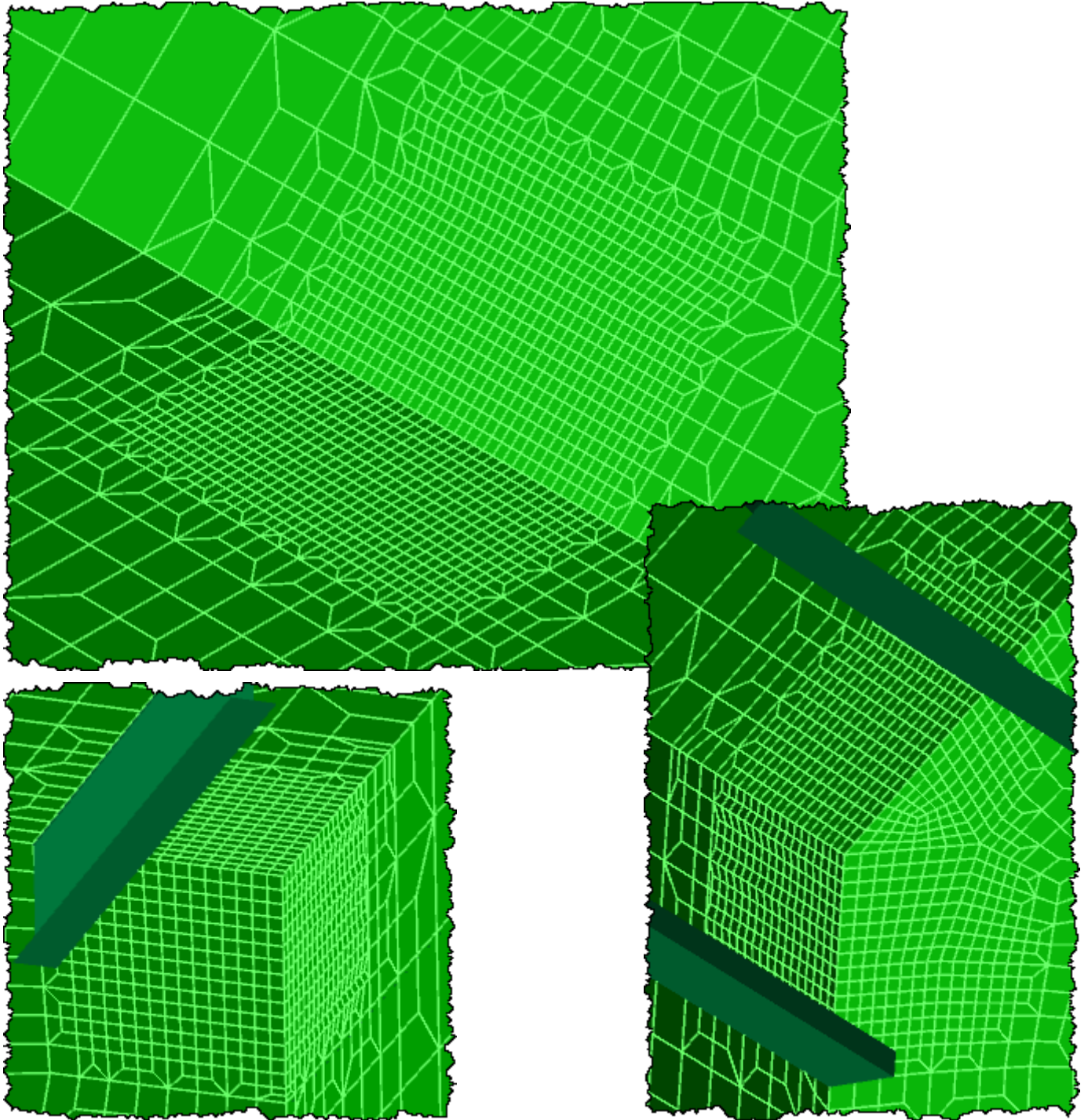


- Prior to clicking *OK* or *Apply*, switch to *Mesh All* display configuration and click *Update Mesh Preview* to see a preview of the mesh refinement.



- Rotate and zoom as desired to study the preview of the mesh refinement.
- Click *OK* to confirm and perform the refinement.

- The resulting mesh within the refinement box is shown below from different angles.



- Note that all mesh editing operations including the *Refine Box* feature are performed on an existing mesh. They are, as for other operations in GeniE, logged. This means that reading the logged commands into a new workspace will redo also the mesh editing operations.
 - But, recreating the mesh will remove the edits!
 - This means that all mesh edit operations must be done as a final step after having run the meshing activity.



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.