

SESAM TUTORIAL

GeniE

Manual Mesh Editing

Valid from program version 8.2





Sesam Tutorial

GeniE – Manual Mesh Editing

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

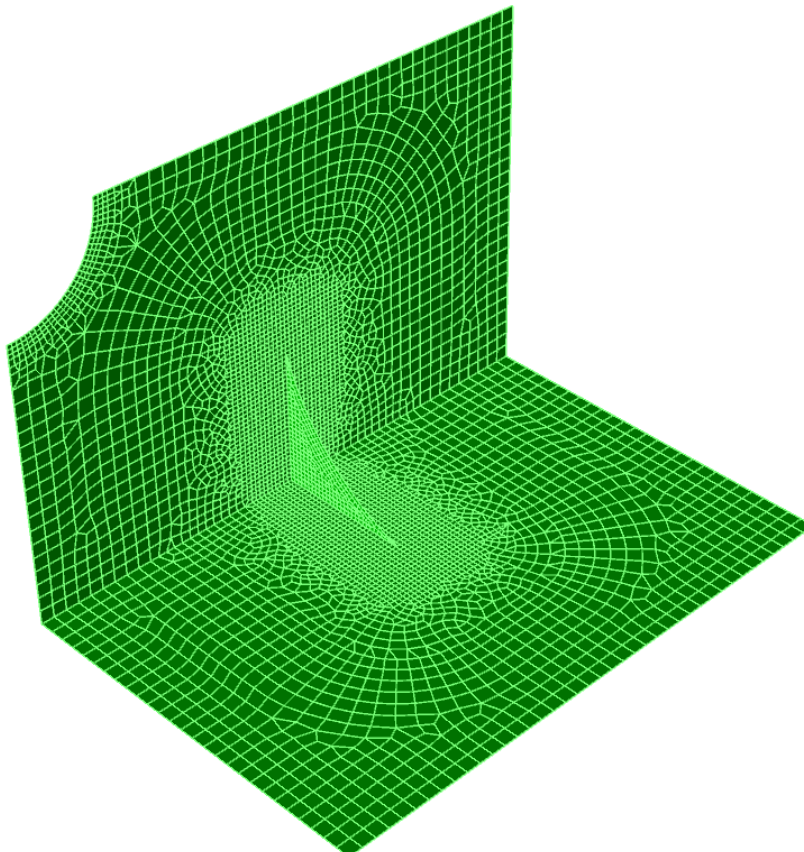
This publication or parts thereof may not be reproduced or transmitted in any form or by any means, including copying or recording, without the prior written consent of DNV AS.

TABLE OF CONTENTS

1. Introduction	Page 4
2. Create a New Workspace and Import a Model	Page 5
3. Assign Mesh Properties	Page 6
4. Create Mesh for Set Named 'fine'	Page 7
5. Edit Mesh Operations Remove Element and Split Edge	Page 9
6. Edit Mesh Operations Imprint Node, Manipulate Triangle and More	Page 12
7. Create Mesh for Set Named 'coarse'	Page 17
8. Edit Mesh Operation Refine	Page 19
9. Create Mesh for the Remaining Part	Page 24
10. Add a Hole Concept and Create a Mesh Around the Hole	Page 28

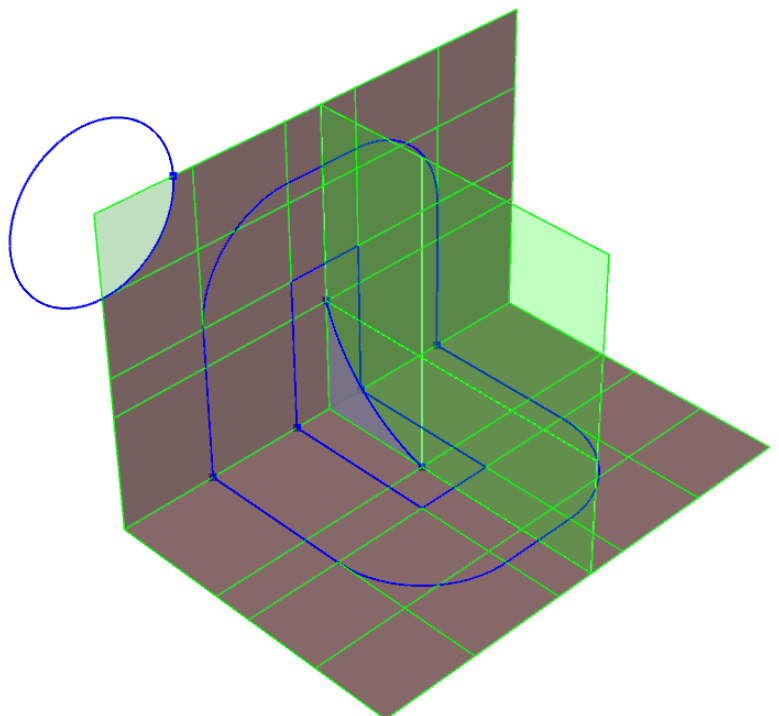
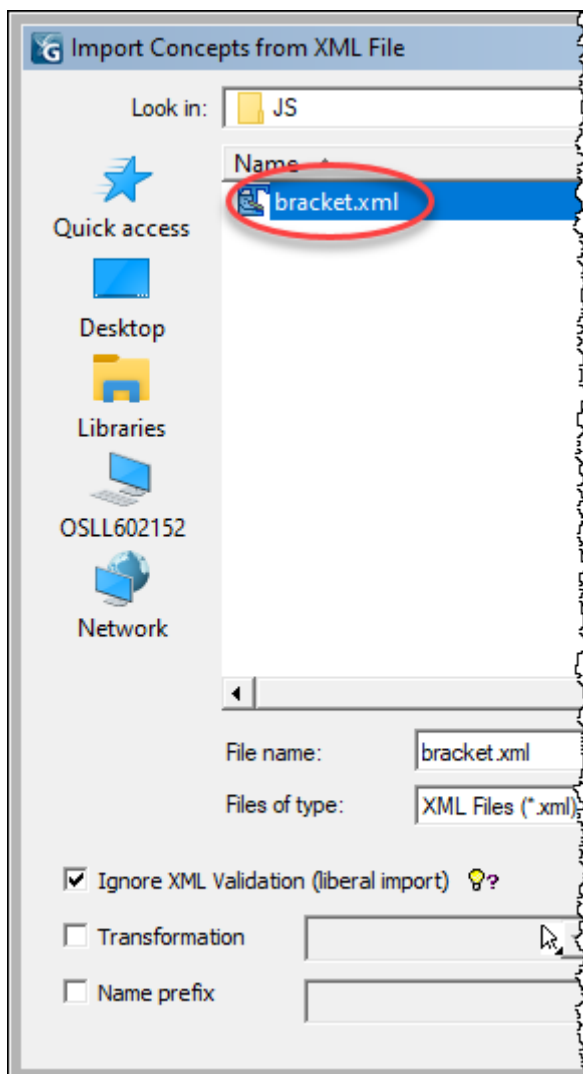
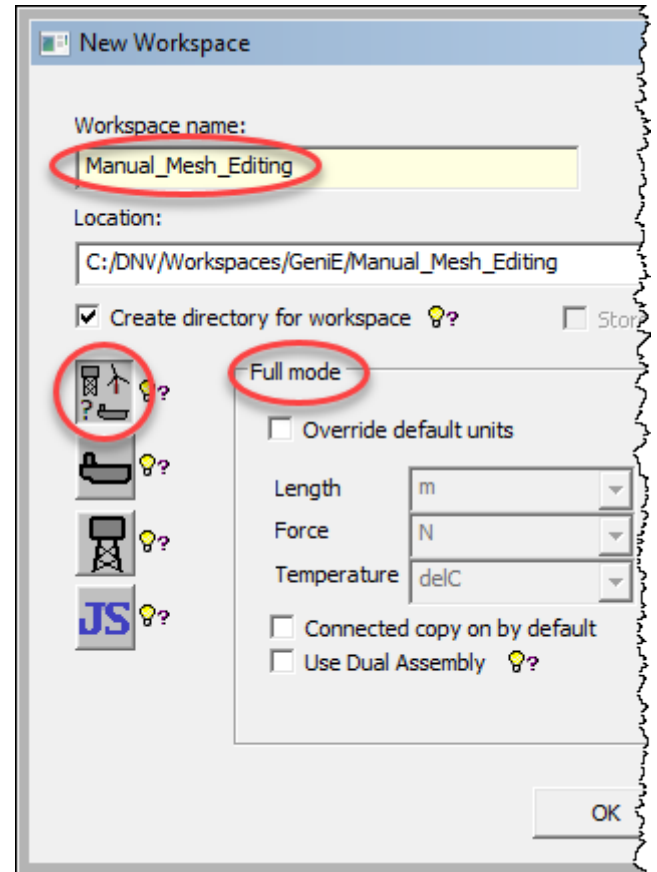
1 INTRODUCTION

- GeniE offers automatic meshing algorithms with a variety of rules, settings, properties and other features for controlling the mesh generation. In most cases, this is sufficient for generating a high quality FE mesh.
- Yet, in certain cases, you may want to have direct control over the mesh, down to the level of modifying individual nodes and elements. This may be when:
 1. An imported mesh needs repair or improvements
 2. The automatically created mesh requires fine-tuning
 3. Rather than including details in the concept model, mesh editing is used to include such details
- The mesh editing presented in this tutorial is for educational purposes and does not necessarily represent cases when the automatic meshing fails in creating a high quality mesh.
- Note the following basic features related to mesh editing:
 - Mesh editing is bound by the underlying concept model. That is, modified nodes and elements will remain on the edges and surfaces to which they belong.
 - While editing, snapping to certain mid-points and nodes eases the process.
 - Subset meshing allows finetuning meshes for parts that need special attention. Such meshes are kept during automatic meshing of the remaining model.



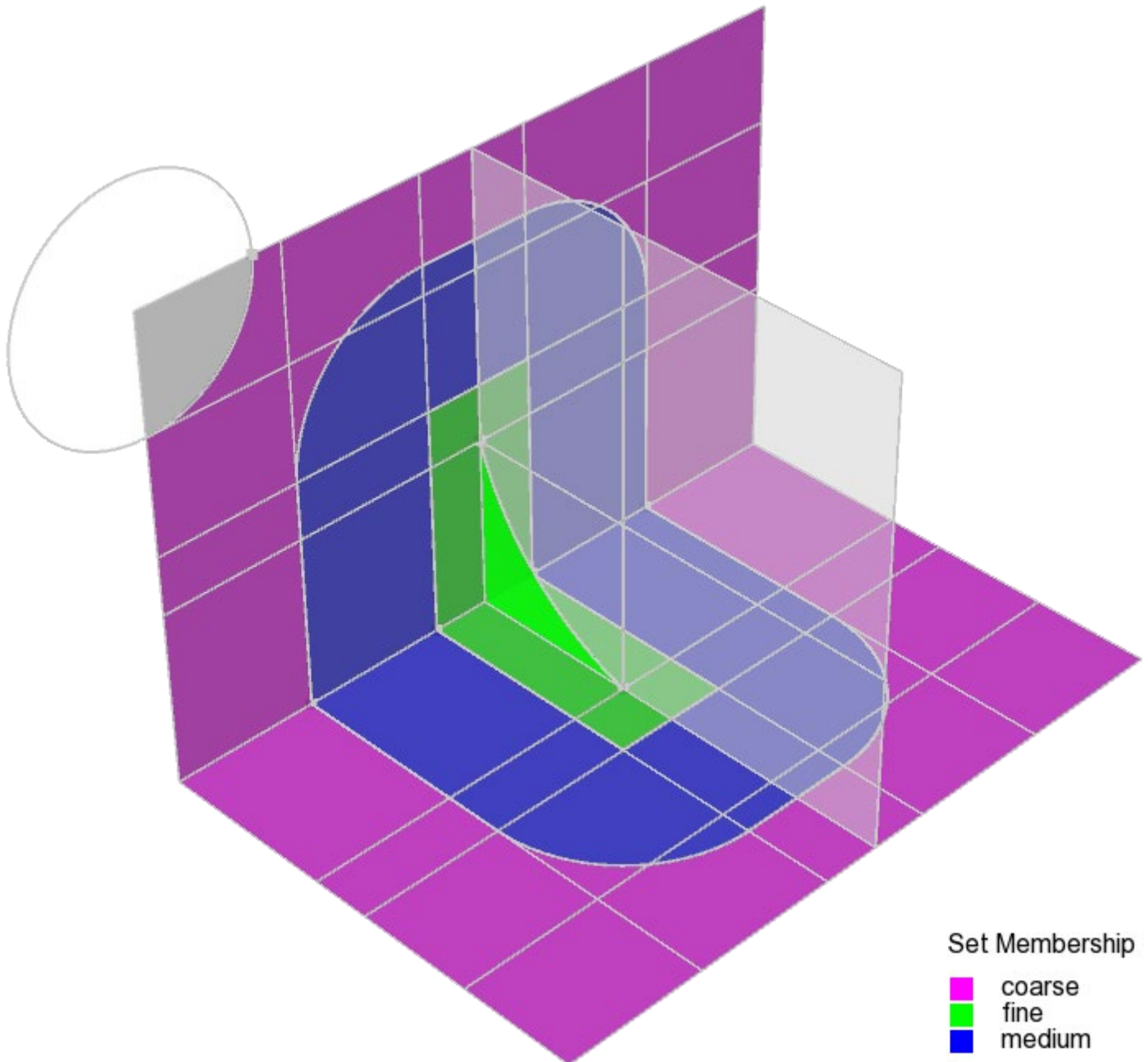
2 CREATE A NEW WORKSPACE AND IMPORT A MODEL

- Start GeniE and open a new workspace.
 - Give a workspace name, for example Manual_Mesh_Editing.
 - 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 bracket.xml. The file is found as part of the installation, typically at:
C:\Program Files\DNV\GeniE VX.Y-ZZ\Help\Tutorials\TutorialsAdvancedModelling\A12_GeniE_Manual_Mesh_Editing\JS
- The model shown to the lower right appears.



3 ASSIGN MESH PROPERTIES

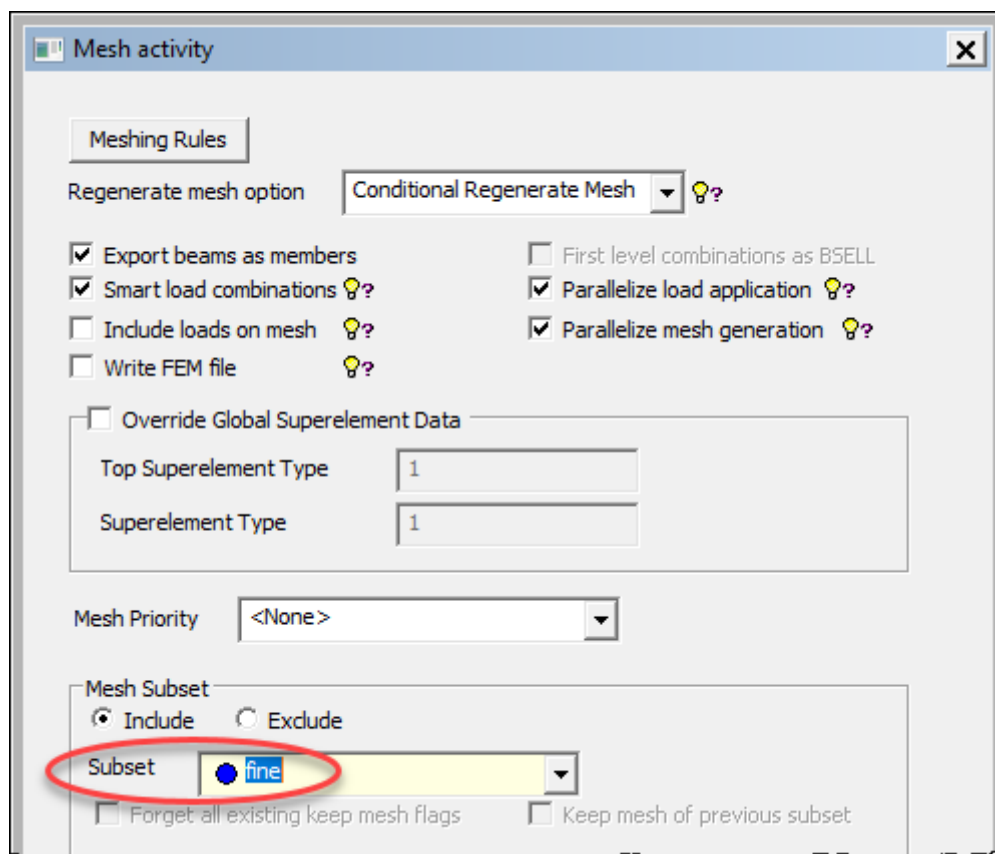
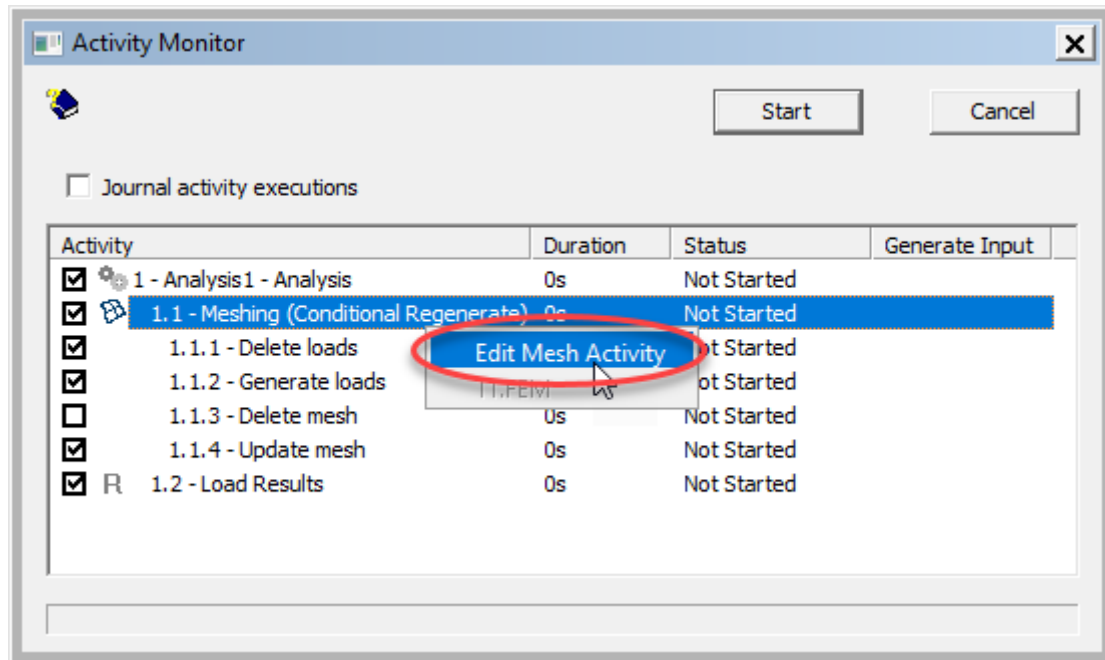
- Three sets have already been defined for this model. They are listed in the *Utilities | Sets | Regular Sets* folder in the browser. The set memberships are colour coded below.



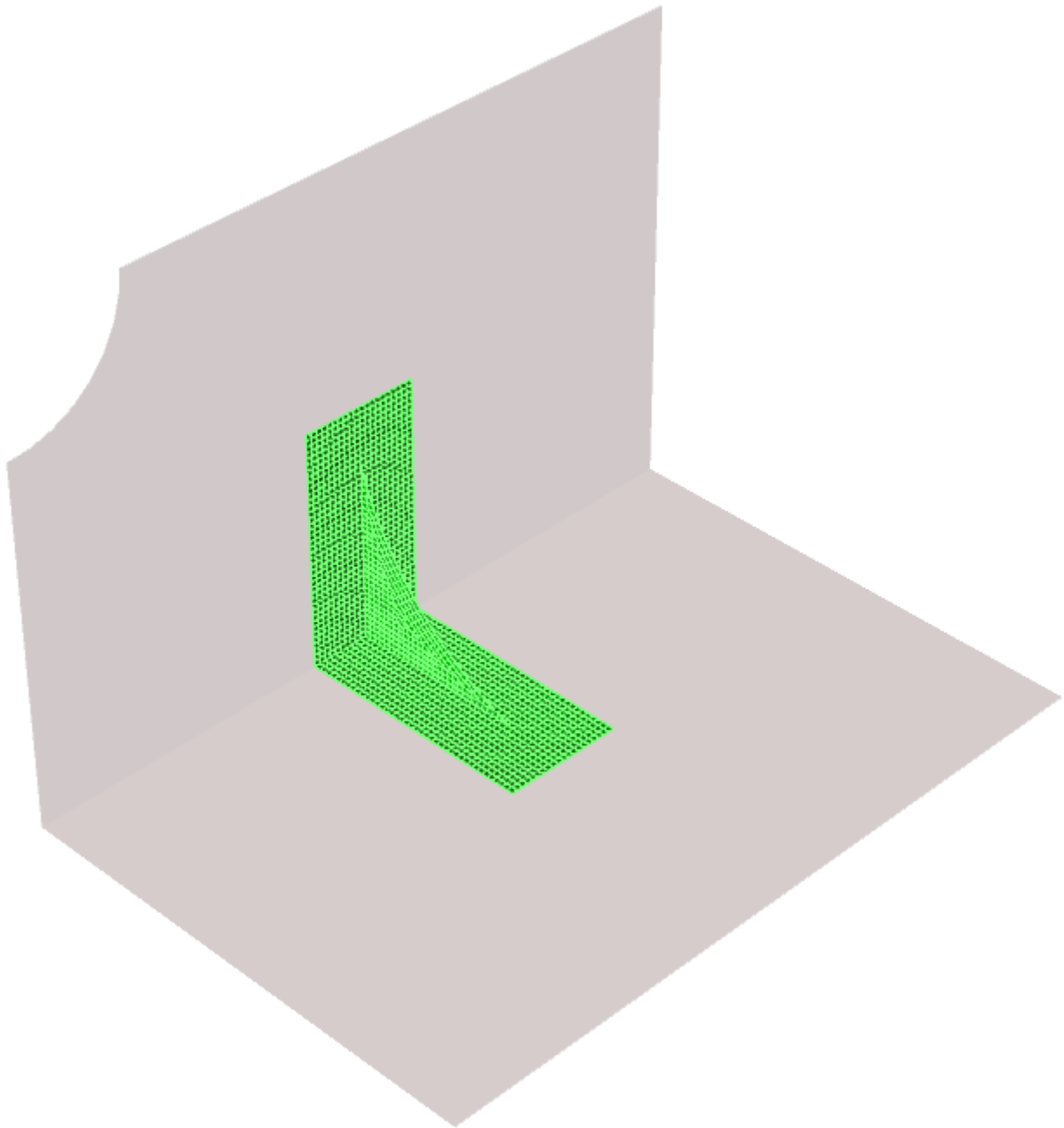
- Assign the following mesh densities to the sets:
 - fine: mesh050
 - medium: mesh100
- Note that there is also a mesh density named 'mesh150' which is set as default mesh property. This means that it will apply to surfaces for which no mesh property has been specifically assigned, i.e. the set named 'coarse'.

4 CREATE MESH FOR SET NAMED 'FINE'

- Rather than meshing the entire model we will fine-tune the mesh for the set named 'fine'.
 - Use Alt+M to open the *Activity Monitor* for the existing analysis activity.
 - Edit the meshing activity and in the *Mesh activity* dialog select *Subset* 'fine'.
 - Thereafter, click *Start* in the *Activity Monitor* to create a mesh.

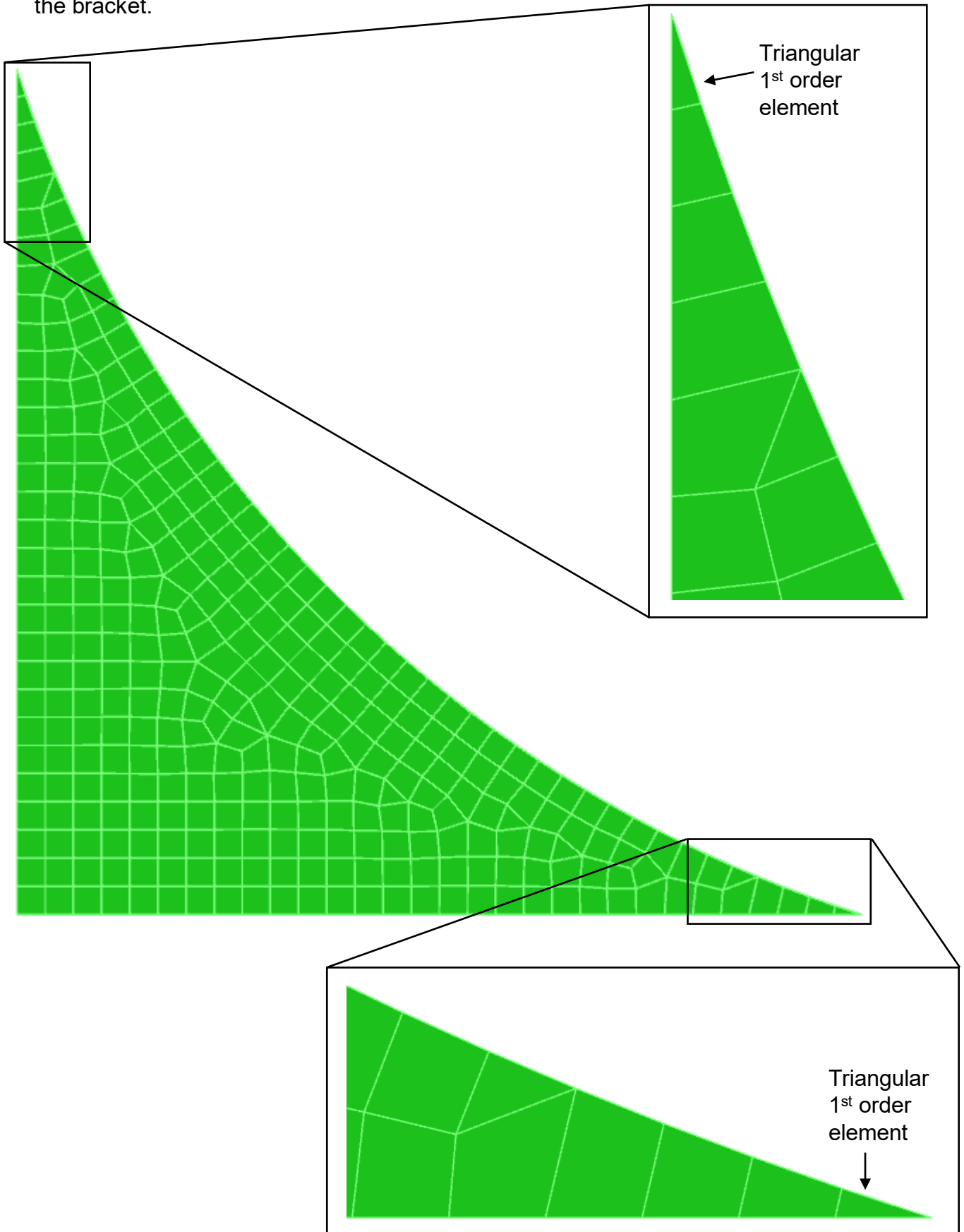


- See that a mesh has been created only for the set named 'fine'.

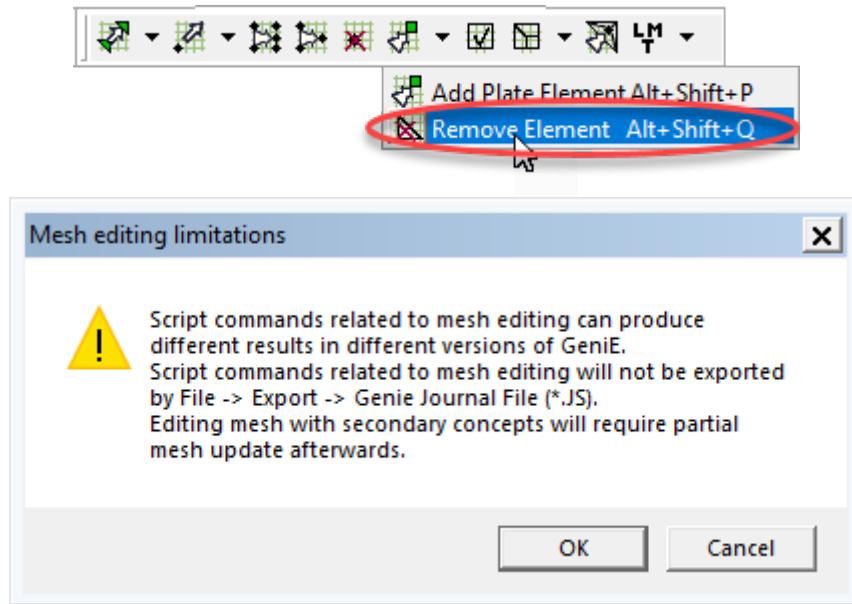


5 EDIT MESH OPERATIONS REMOVE ELEMENT AND SPLIT EDGE

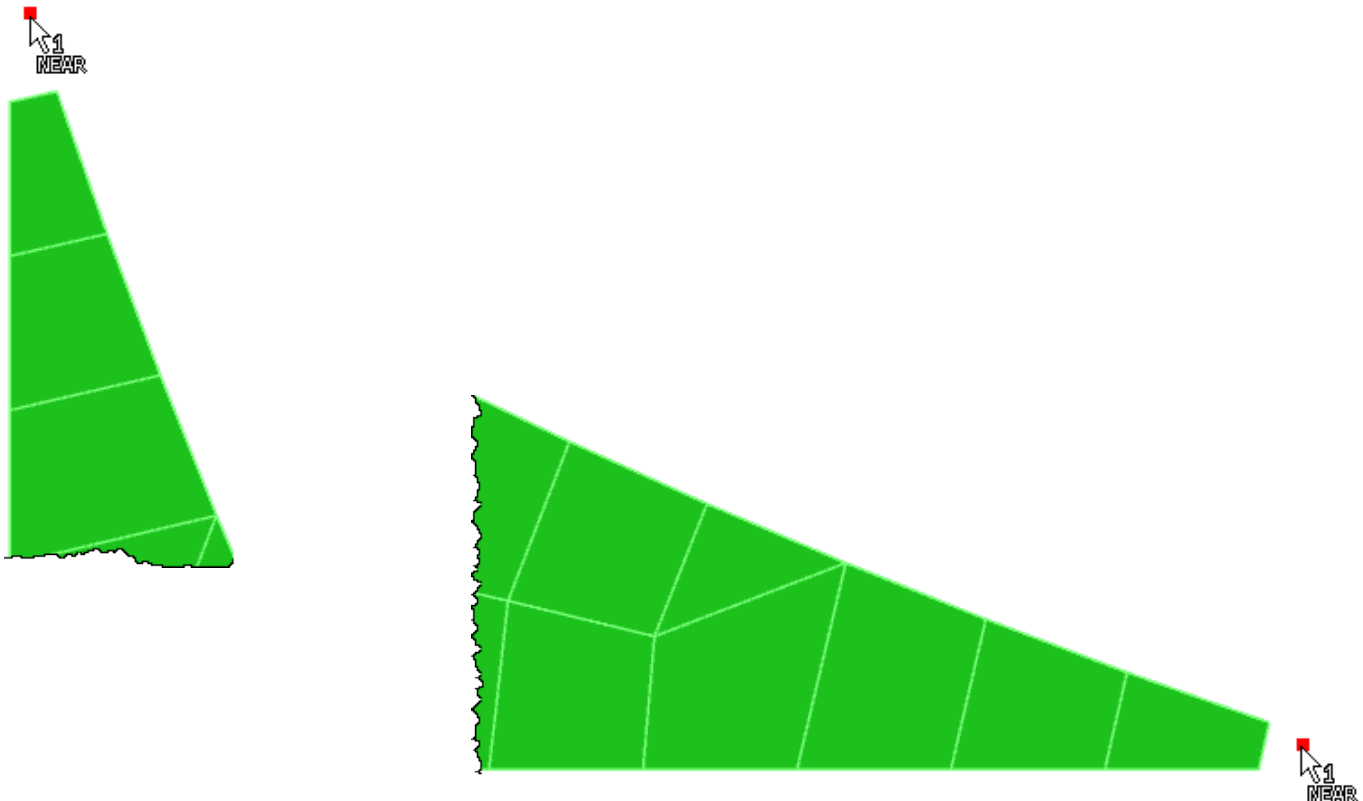
- View the bracket only and notice the triangular 1st order elements in the pointed corners of the bracket.



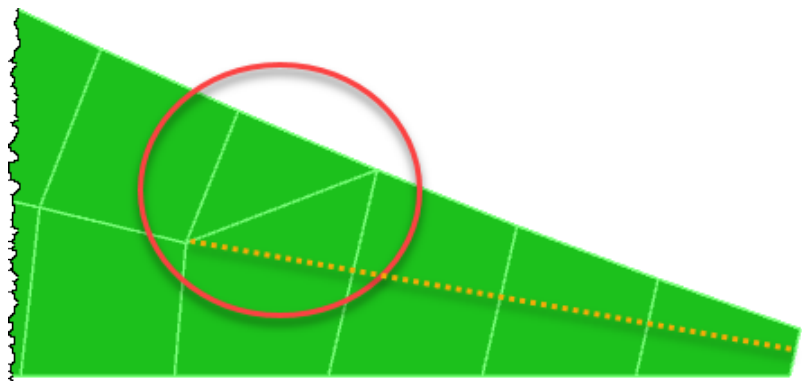
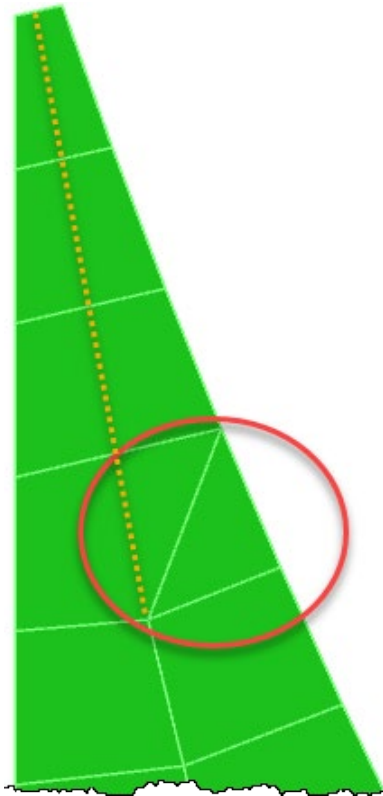
- 1st order triangular elements tend to be very stiff. This can lead to unsatisfactory analysis results when they occur in highly stressed areas like the pointed ends of the bracket. A simple remedy is to delete these triangular elements.
- Select the tool button *Remove Element*. The *Mesh dialog limitations* dialog appears, click *OK* to the message. (This message only appears once.)



- See that when hovering the mouse over an element it disappears from the display. Click on the two triangular elements at the pointed ends of the bracket to remove them.

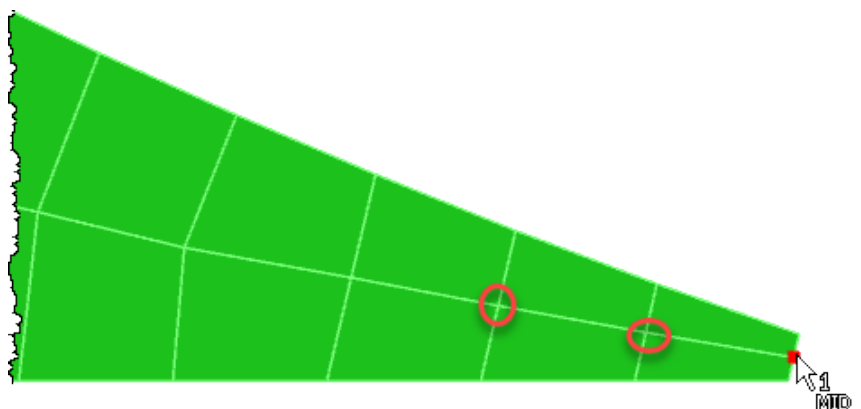
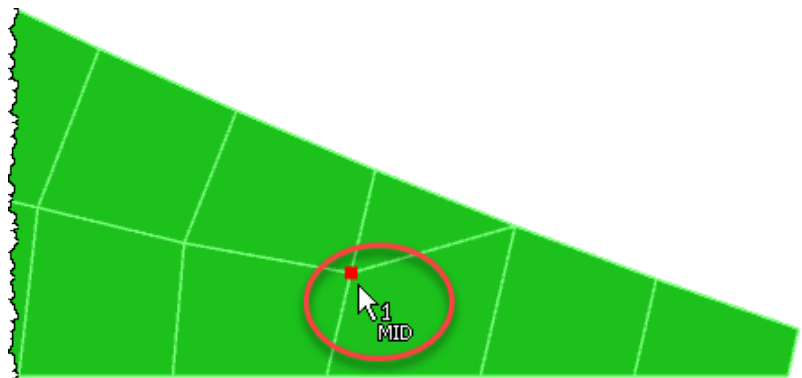


- There is still triangular elements near the pointed ends of the bracket:



- These may be removed by splitting the elements as indicated by the orange dotted lines above.

- Do so by pressing the *Split edge* tool button. When hovering as shown to the right the tool tip *MID* appears indicating that when clicking a new mesh line will split the near vertical mesh line at its midpoint.
- Continue clicking to split elements as shown to the right.
- Do the same for the upper pointed end.



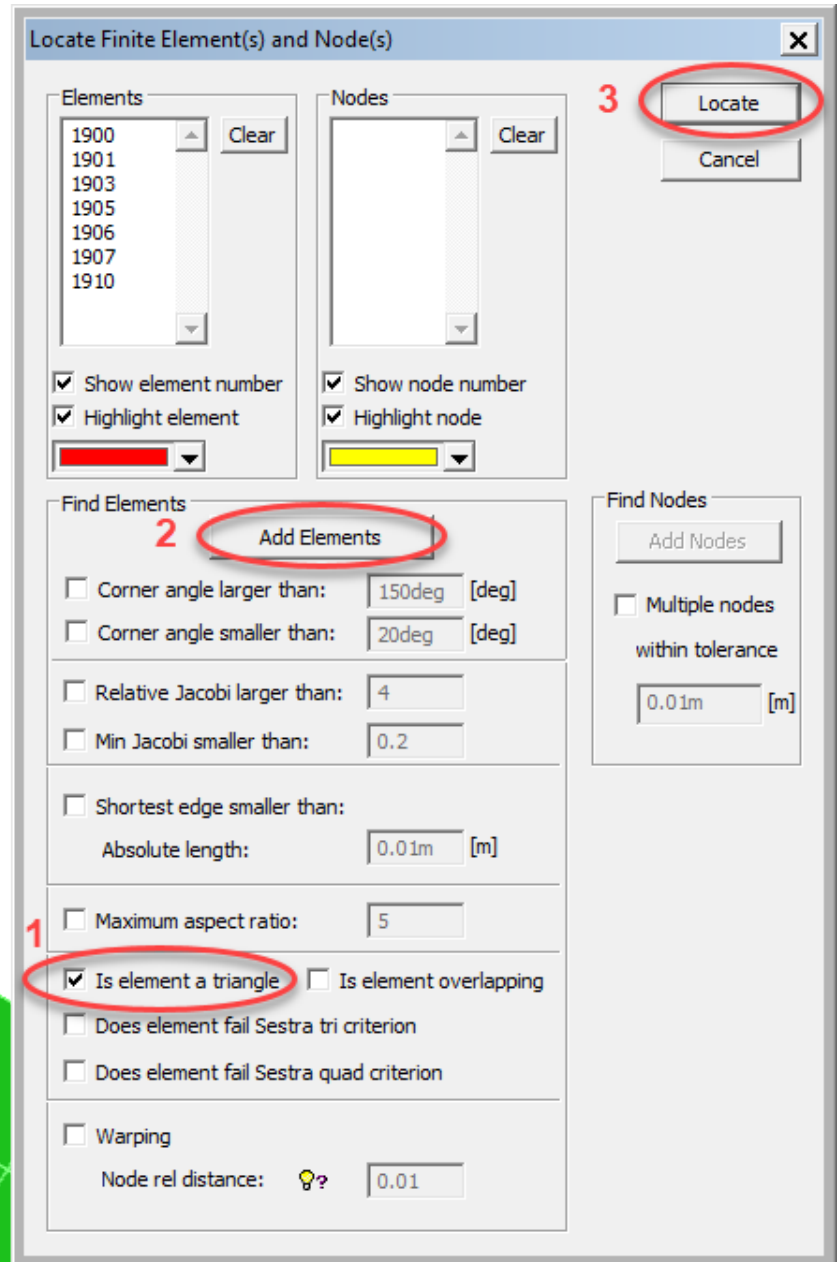
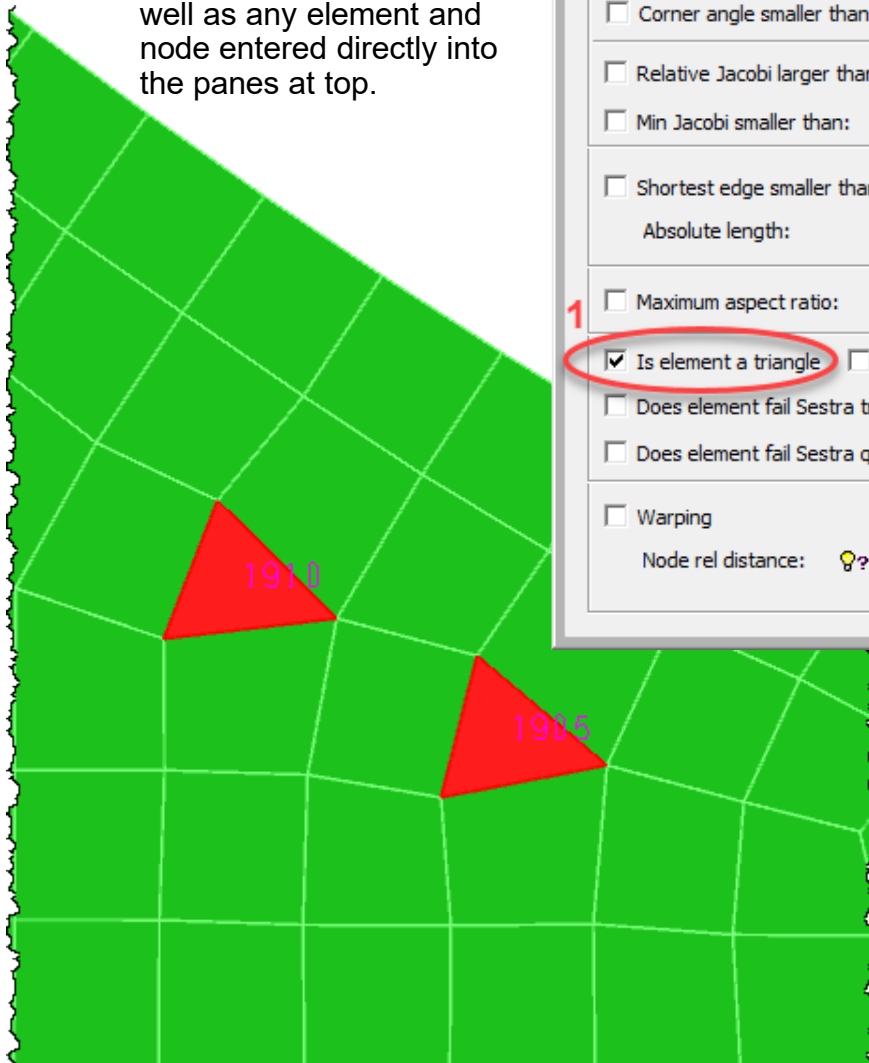
6 EDIT MESH OPERATIONS IMPRINT NODE, MANIPULATE TRIANGLE AND MORE

- There are more triangular elements in the bracket. These are of less consequence but as an exercise let us remove some of them. To do so may require some imagination plus practising through trial and error. Remember that Ctrl+Z can be used to undo actions.

- Let us focus on the highlighted triangles near the lower pointed corner.

- Triangles may be highlighted by *Mesh & Analysis | Locate Finite Element*. In the *Locate Finite Element(s) and Node(s)* dialog, (1) check *Is element a triangle* followed by (2) clicking *Add Elements* and finally (3) *Locate*.

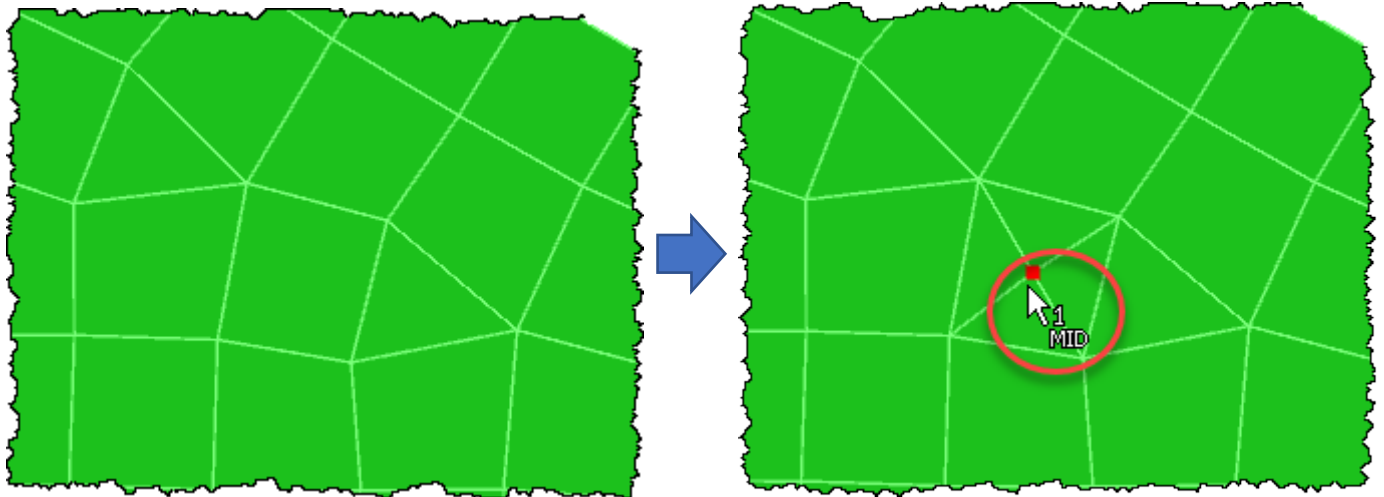
- Notice that this dialog also offers locating and highlighting elements with various imperfections as well as any element and node entered directly into the panes at top.



➤ Press the *Imprint Node* button and hover the mouse as shown to the right below.

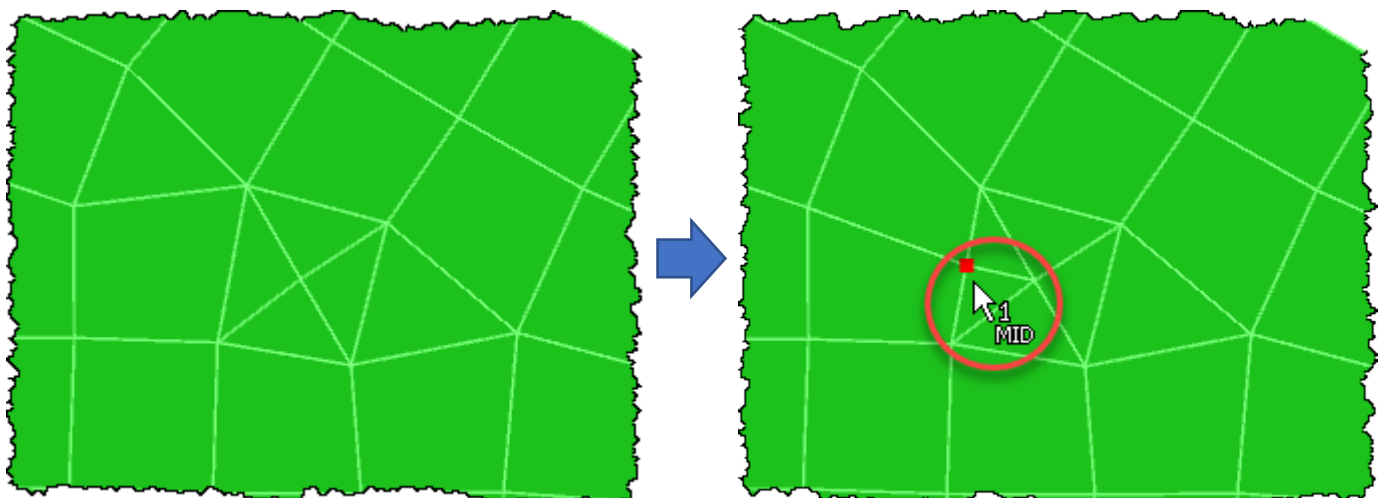
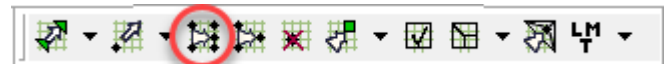


- Ensure the tool tip says *MID* so that when clicking, the new node is positioned in the middle of the original quadrilateral element.
- See that the original quadrilateral element is split into four triangular elements.



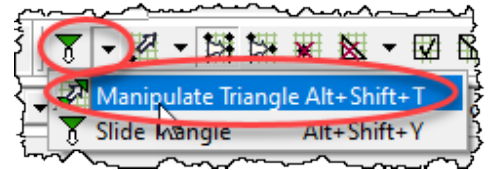
➤ At this point four *new* triangles have been added and no triangles removed. This is where the imagination plus practicing through trial and error comes in.

➤ Press the *Split edge* button and hover the mouse as shown to the right below.

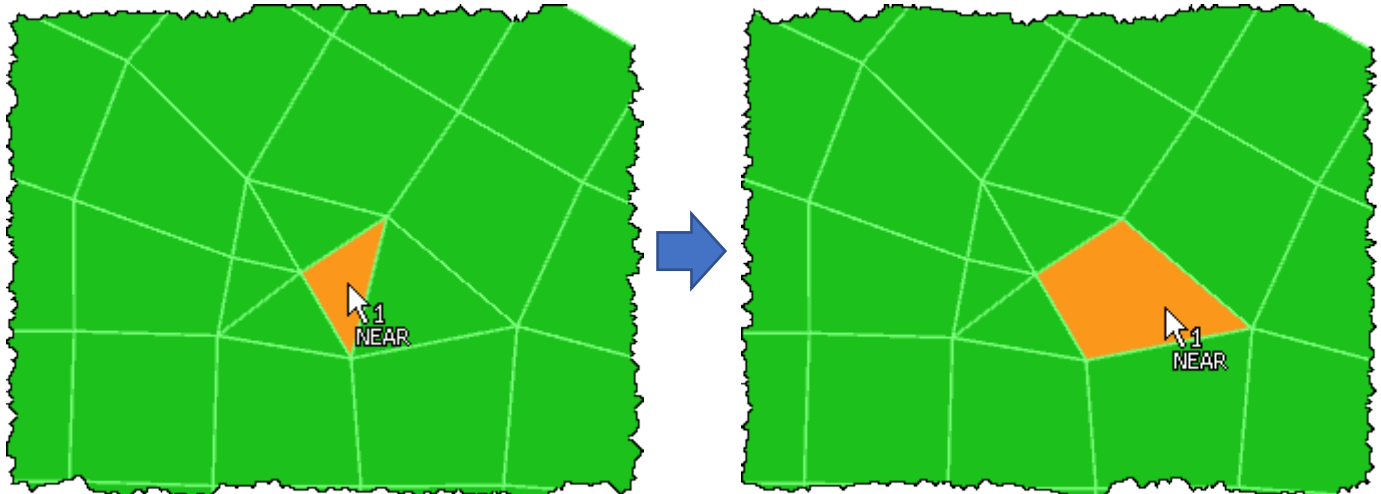


➤ At this point one of the original triangular elements have been removed. But still there are several triangles.

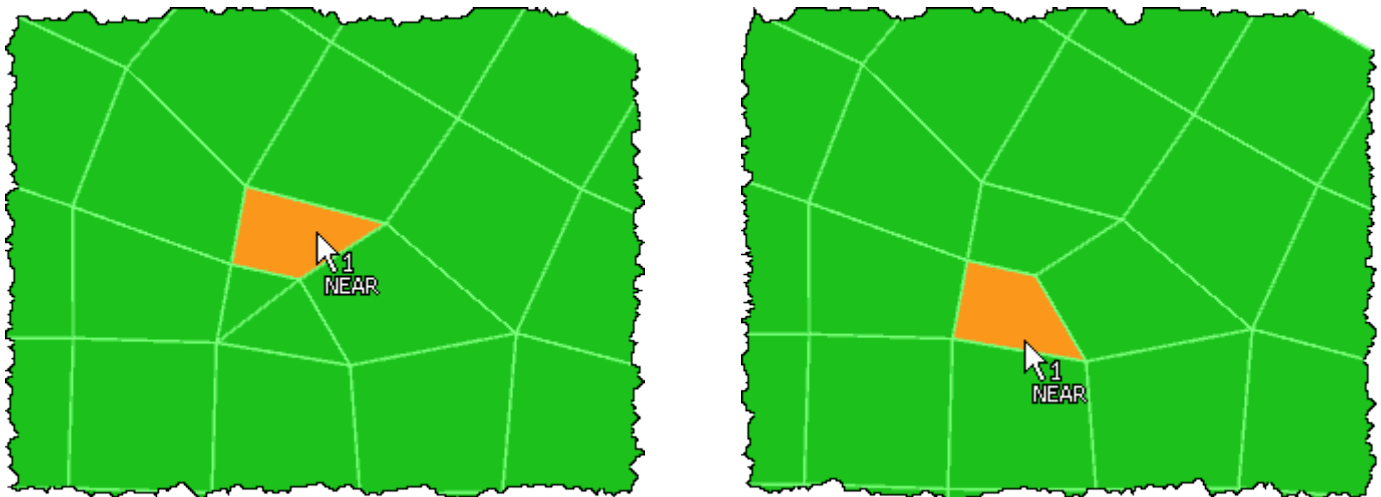
- Press the *Manipulate Triangle* button and hover the mouse over the triangular element as shown to the left below.



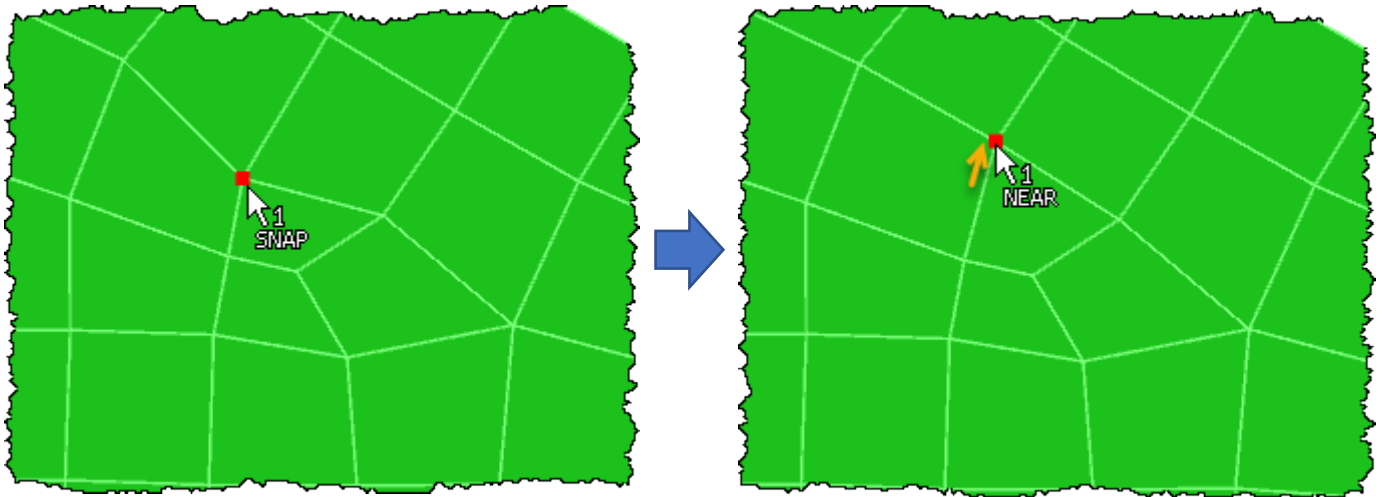
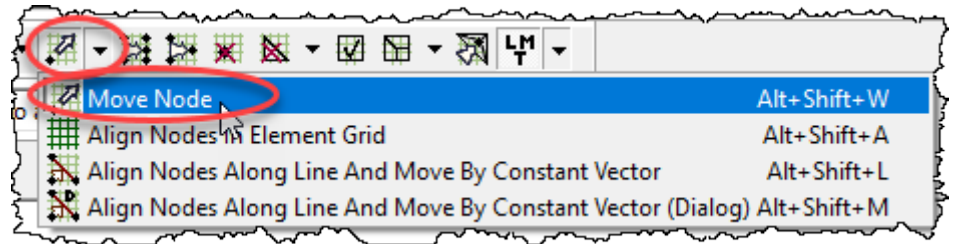
- Press and drag the mouse as shown to the right below.



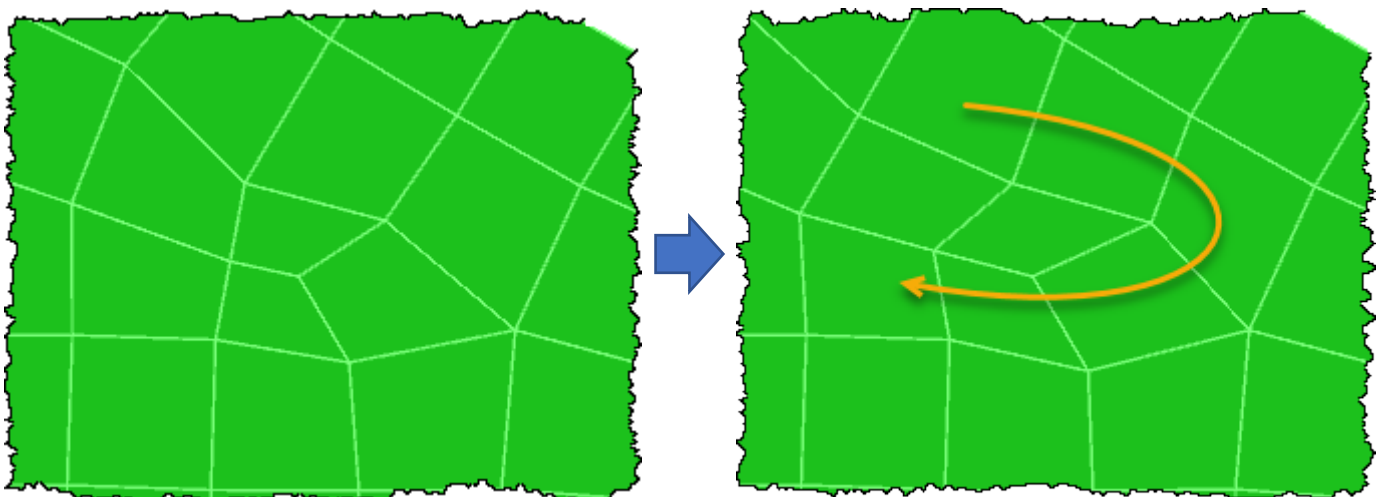
- Join the two sets of remaining triangular elements as shown to the left and right below.



- Further improvements may be done by pressing the *Move Node* button to manually move nodes.



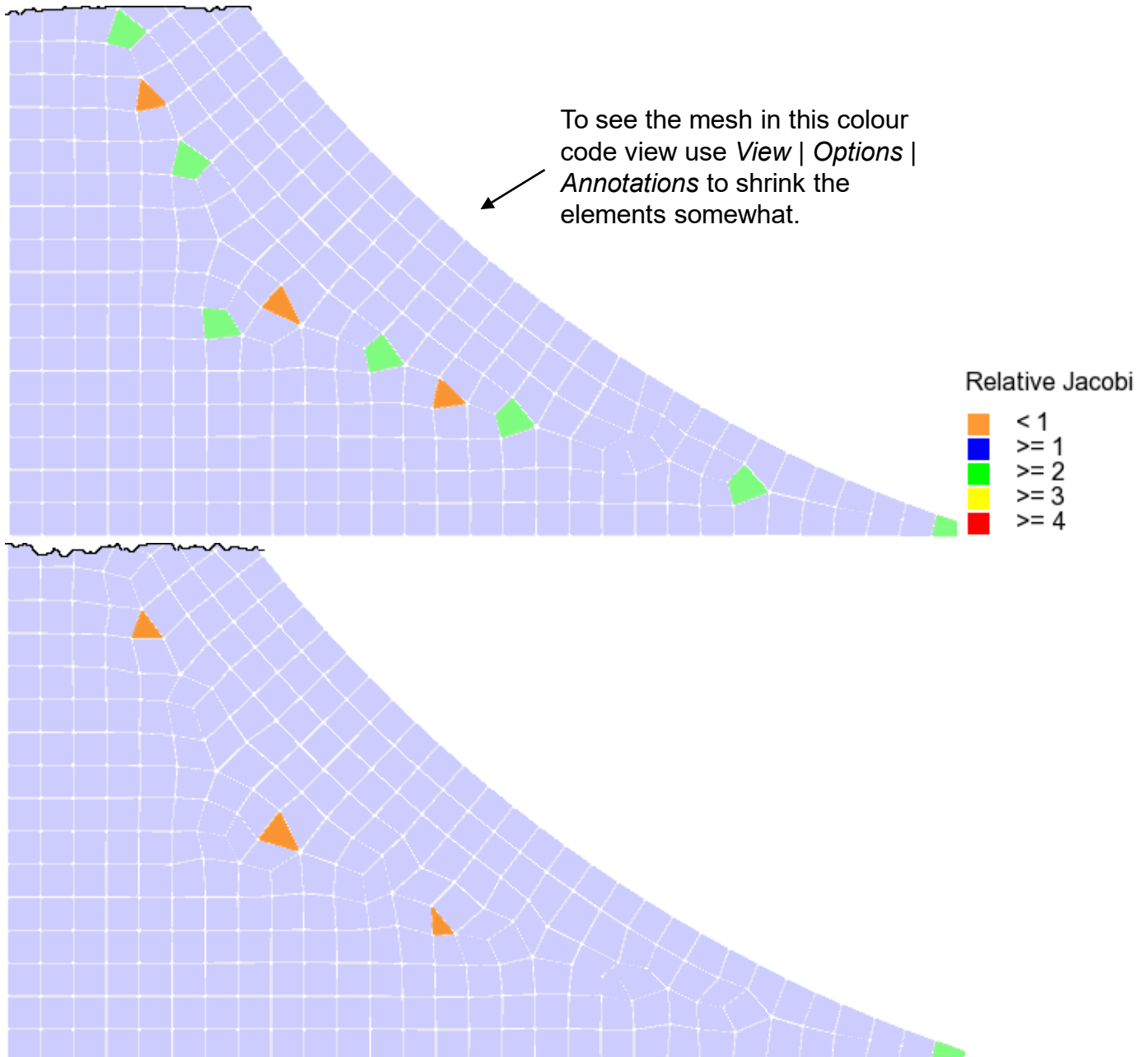
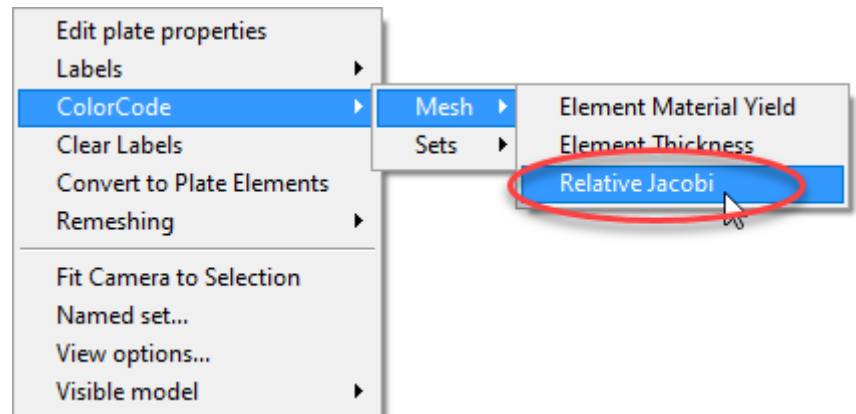
- Overall mesh improvements may also be done by pressing the *Enhance Quality* button and then sweeping the pressed mouse over elements. Nodes are moved during the sweeping process, continuously improving the elements.



- The benefit of the *Enhance Quality* function is seen when using this feature combined with colour coding of the relative Jacobian determinant.

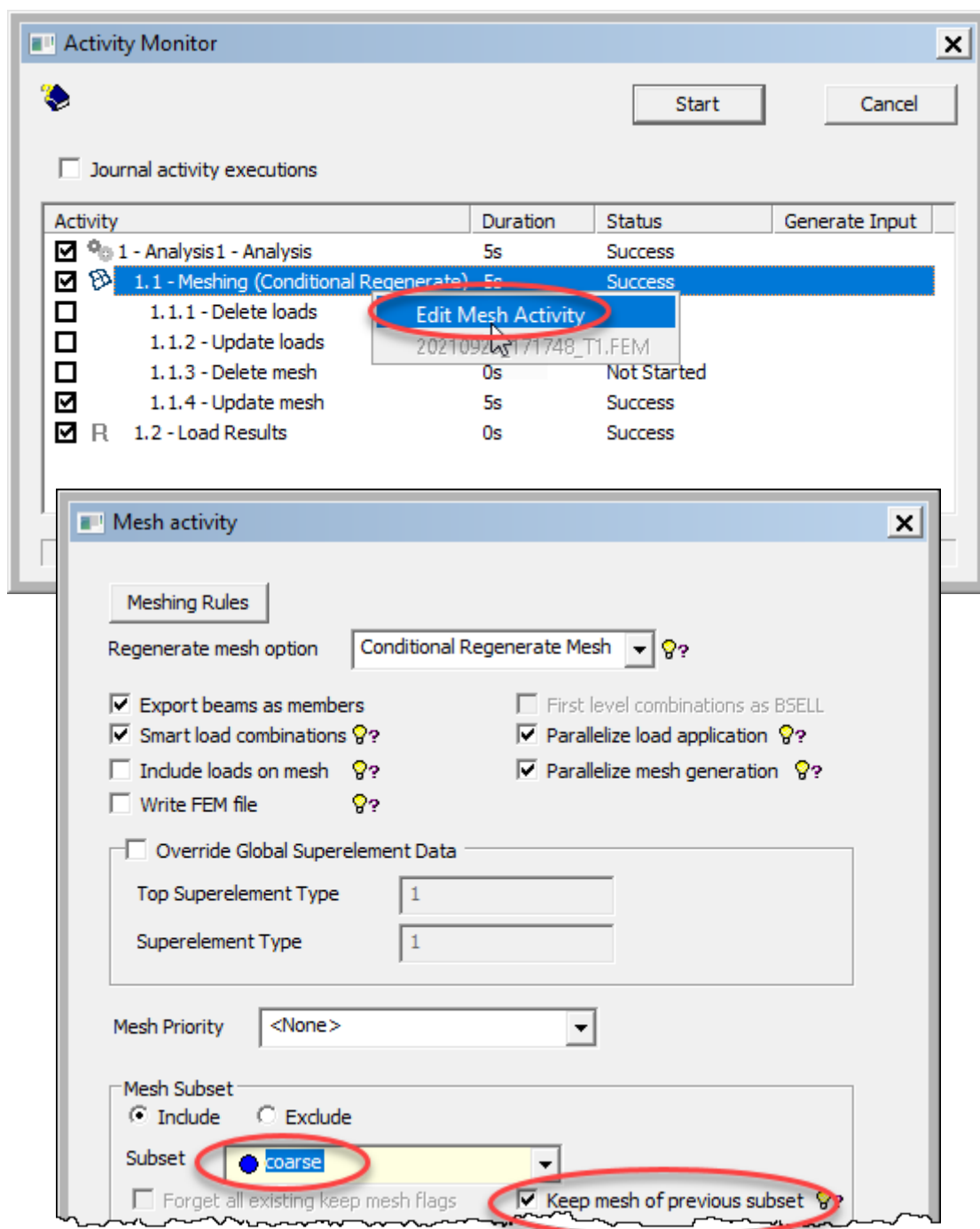
- First, select the whole bracket to colour code the *Relative Jacobi*.

- Below is shown before and after enhancing the quality by sweeping over all green quadrilateral elements, i.e. those with relative Jacobian determinant higher than 2. All quadrilateral elements, except the ones in the pointed corner, end up with determinants less than 2. Triangular elements cannot be improved in this way.



7 CREATE MESH FOR SET NAMED 'COARSE'

- The mesh for the set 'fine' is finalised.
- We will now create a mesh for the set named 'coarse'.
 - Use Alt+M to open the *Activity Monitor* for the existing analysis activity.
 - Edit the meshing activity and in the *Mesh activity* dialog select *Subset* 'coarse'.
 - Make sure the *Keep mesh of previous subset* is checked as we want to keep the mesh for the set 'fine'.
 - Thereafter, click *Start* in the *Activity Monitor* to create a mesh.

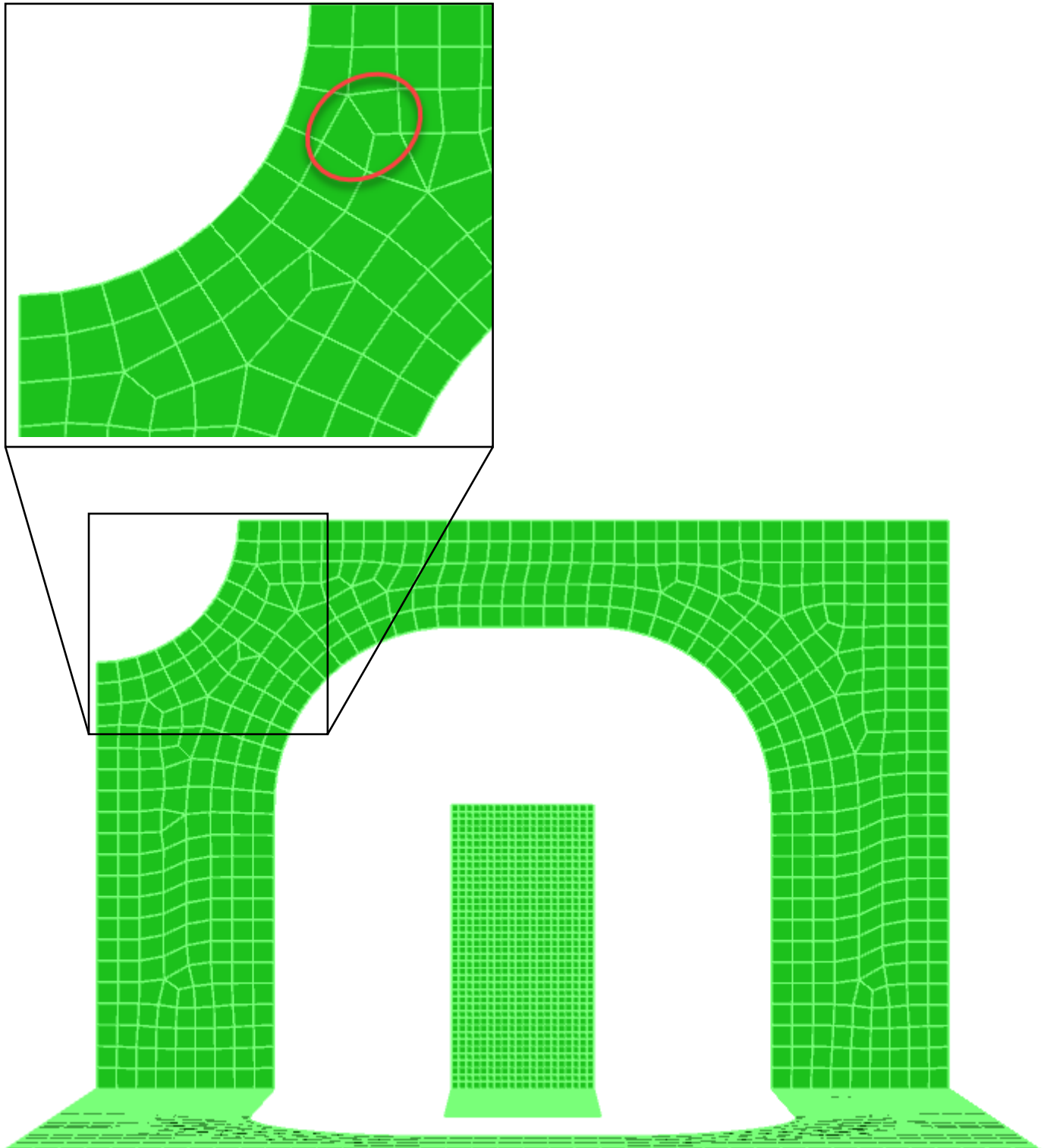


- Using Alt+A to display the whole model and pressing the function key F5 displays the current mesh for the whole model. Also switch off colour coding by lifting the *Property color coding of all visible structure* button.



8 EDIT MESH OPERATION REFINE

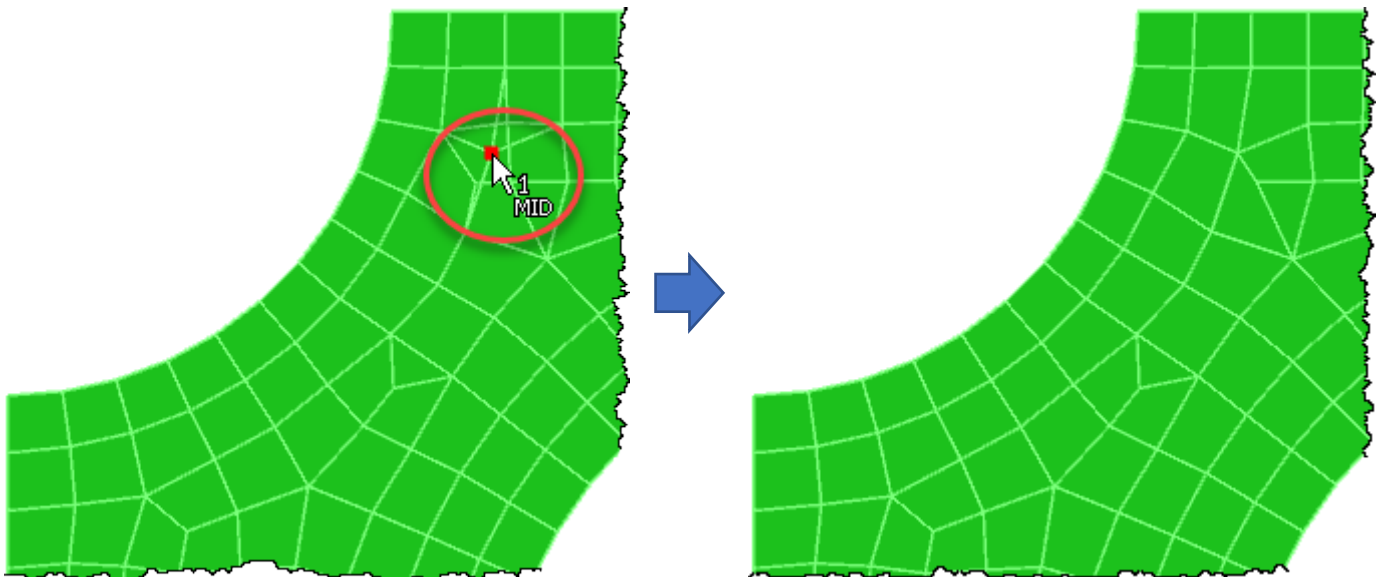
- Let us concentrate on the cutout shown below.
- We want to refine two rows of elements next to the cutout.
 - Notice the two irregular quadrilateral element encircled below.
 - To enable refinement, we want two, more or less, regular rows of quadrilateral elements.



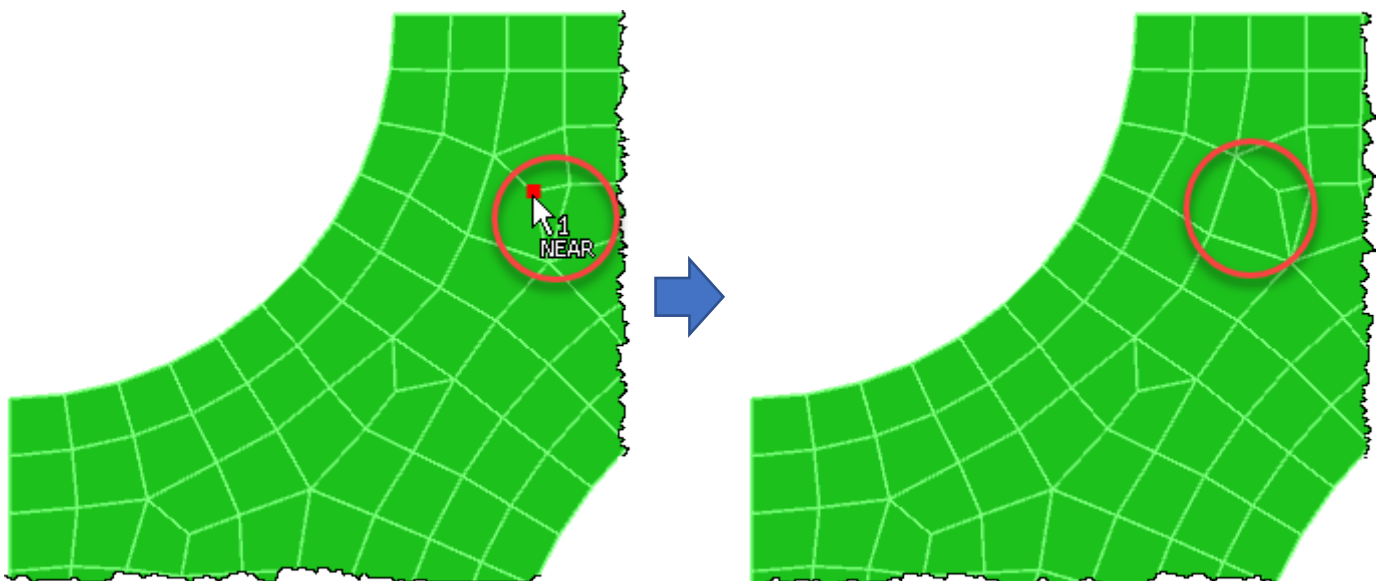
- Press the *Collapse Edge* button and hover the mouse over the quadrilateral element as shown to the left below.



- Click when the tool tip spells *MID*, this creates a new node at the midpoint of the clicked quadrilateral element which is removed and neighbouring elements modified to fill the void as shown to the right below.



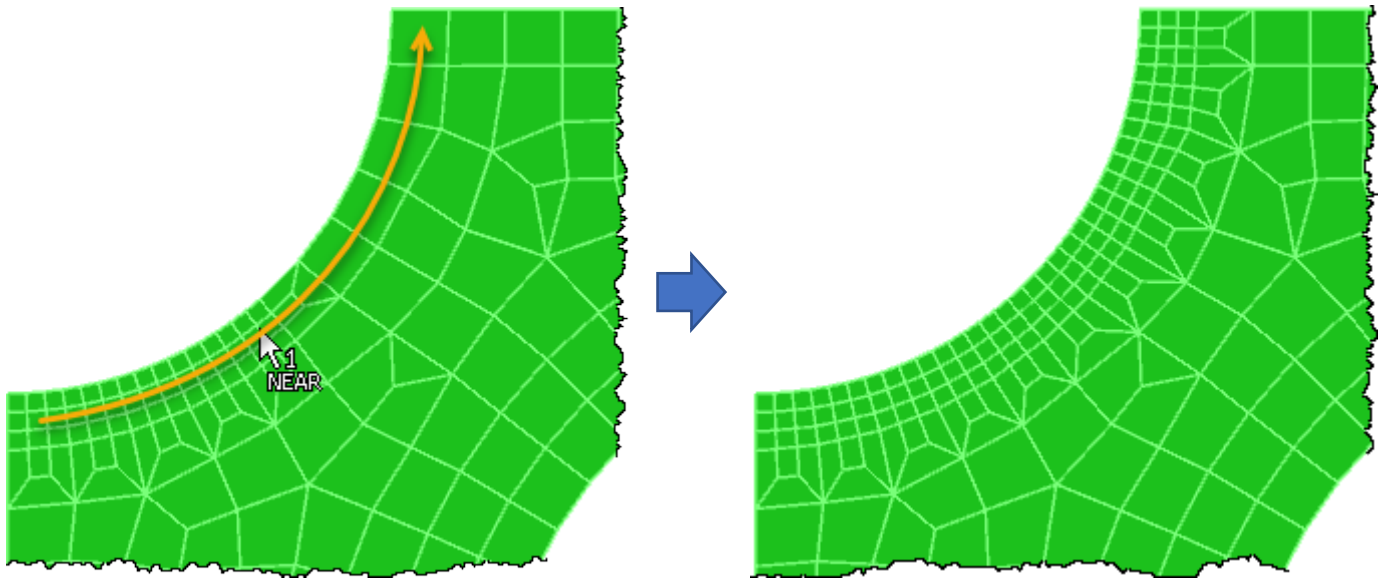
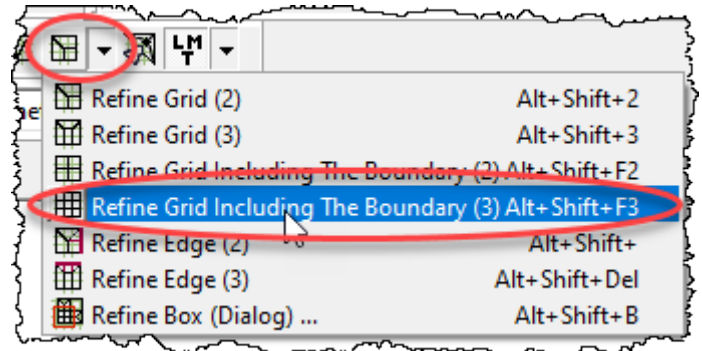
- Use *Move Node* to improve the quadrilateral element encircled to the right below.



- The mesh is now ready for refinement of the two, more or less, regular rows of quadrilateral elements next to the cutout.

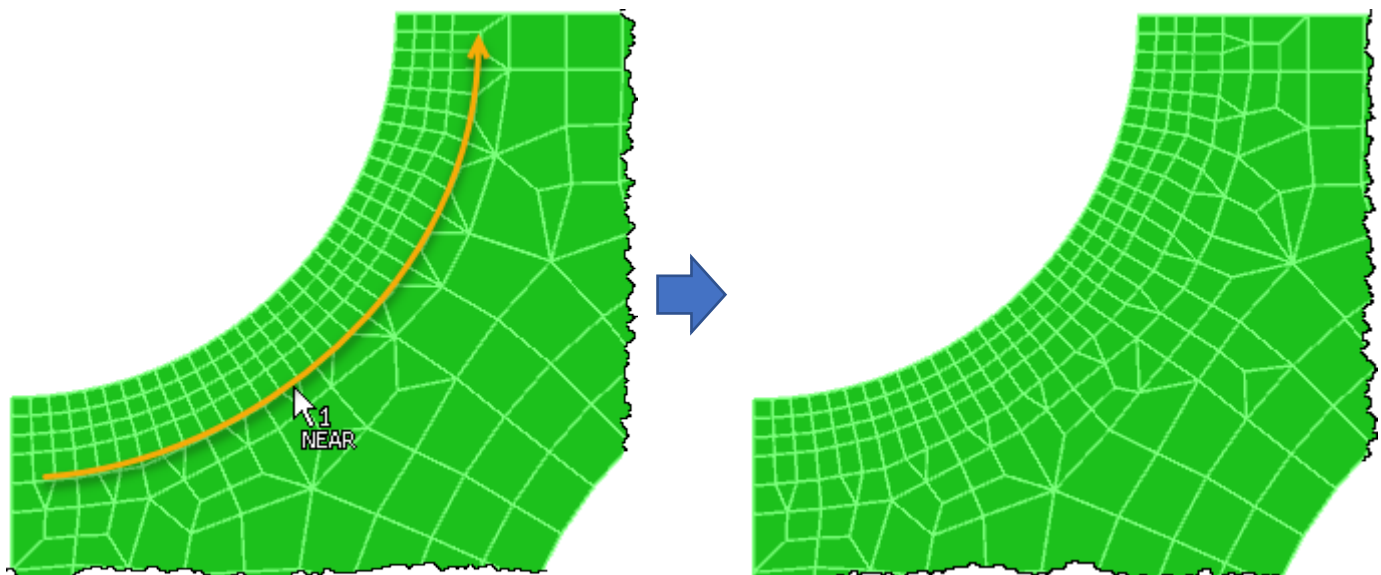
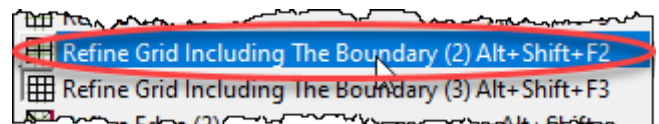
➤ Press the *Refine Grid Including The Boundary (3)* button.

- Sweep the pressed mouse over the first row of elements next to the cutout and see that the mesh is progressively refined. Do not release the mouse until resting over the last element as shown by the orange arrow below.










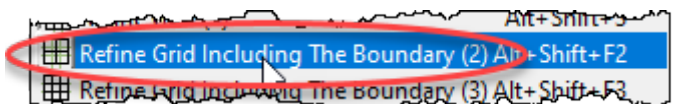
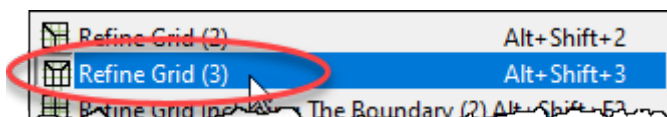
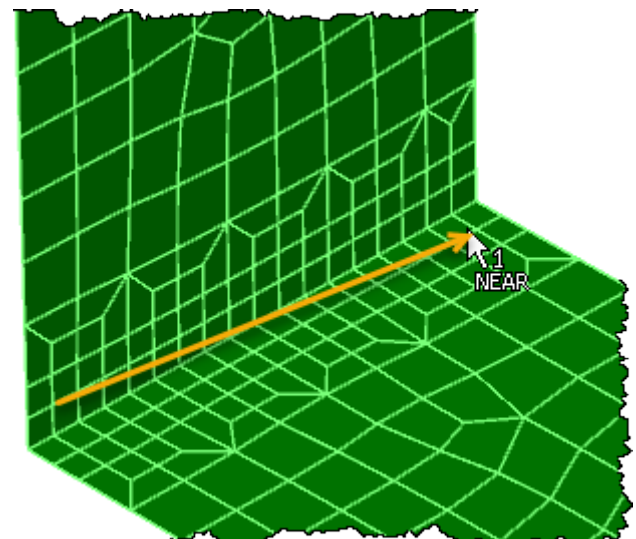
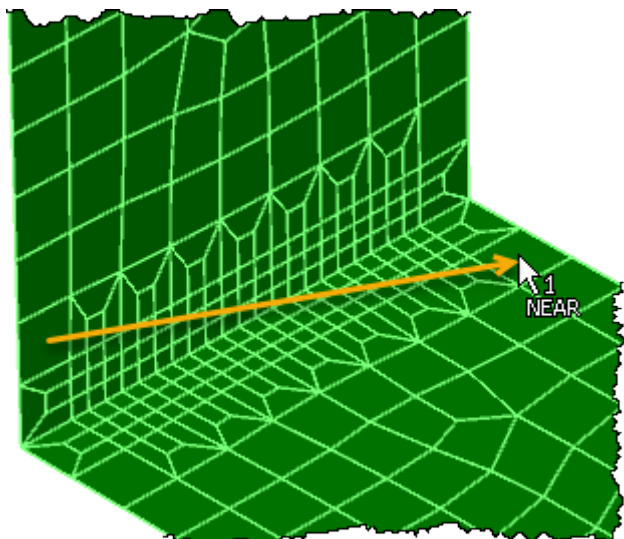
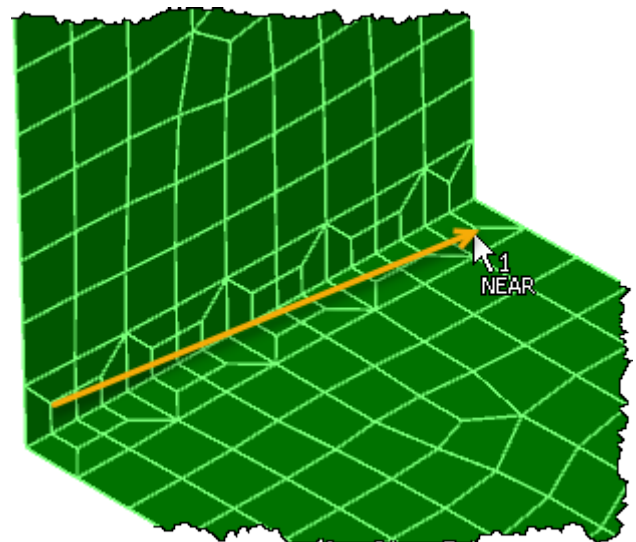
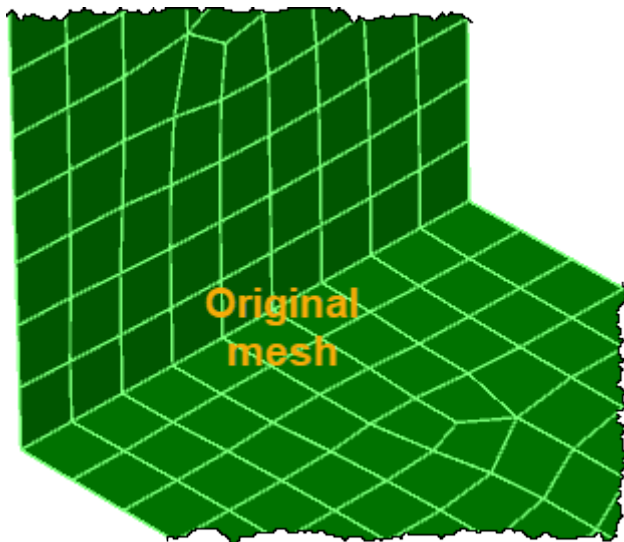
➤ Notice that the original second row becomes a transition zone to the coarser mesh outside. This row is yet open for further refinement.

➤ Press the *Refine Grid Including The Boundary (2)* button and sweep the pressed mouse over the second row of elements as shown below.

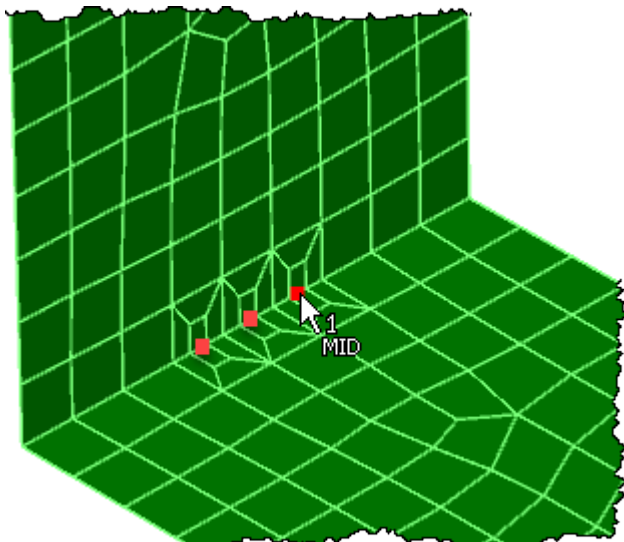
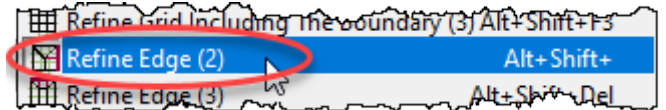
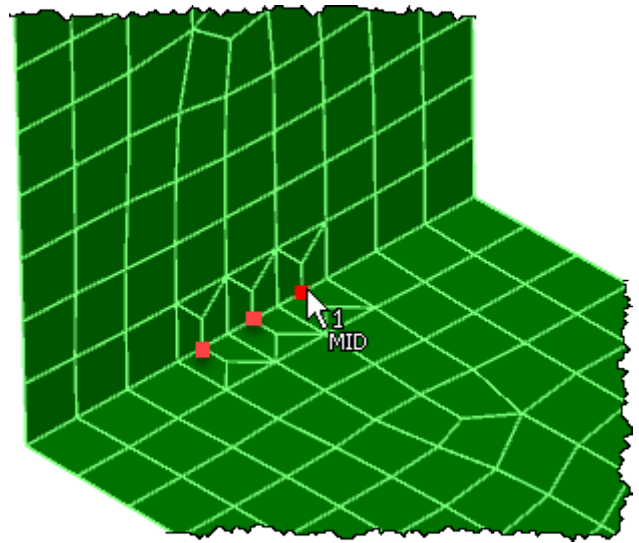
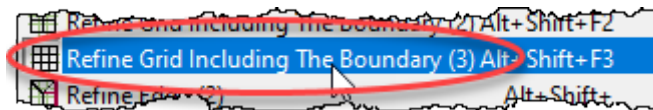
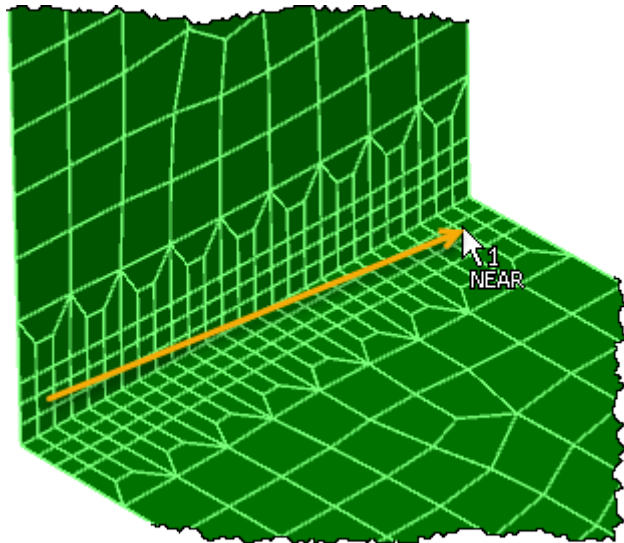


- When the second row is further refined the transition to the coarser overall mesh is pushed further out.
- There are several options for refining the mesh as shown to the right.
- Examples of these are shown below, each reversed by undo (Ctrl+Z).

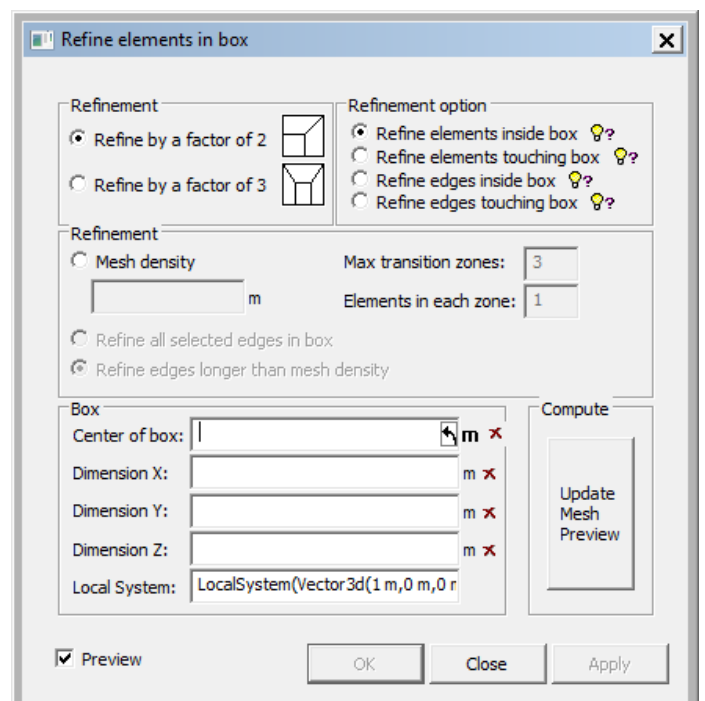
	Refine Grid (2)	Alt+Shift+2
	Refine Grid (3)	Alt+Shift+3
	Refine Grid Including The Boundary (2)	Alt+Shift+F2
	Refine Grid Including The Boundary (3)	Alt+Shift+F3
	Refine Edge (2)	Alt+Shift+
	Refine Edge (3)	Alt+Shift+Del
	Refine Box (Dialog) ...	Alt+Shift+B



- More examples of refinements, each followed by undo (Ctrl+Z).

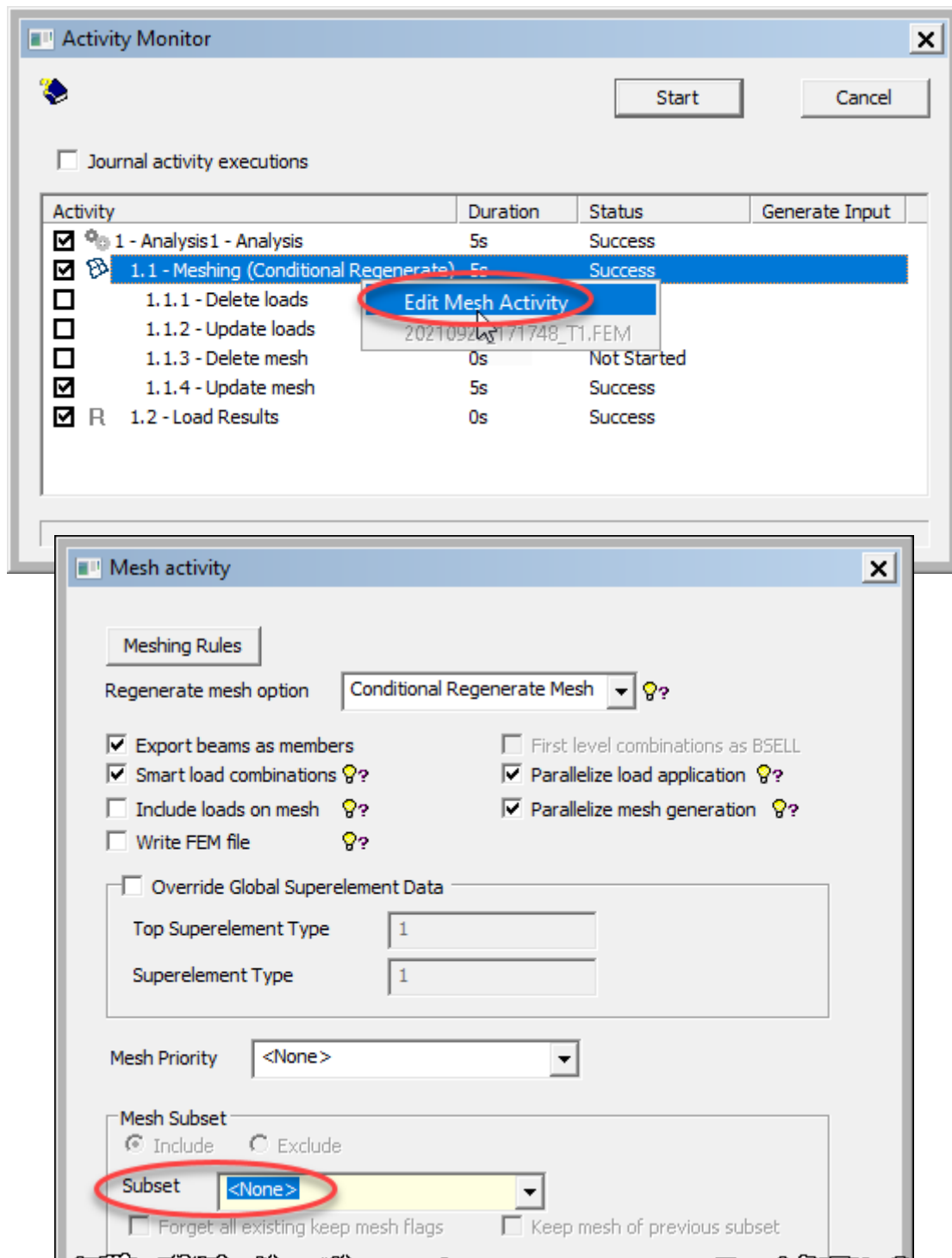


- Additionally, there is a *Refine Box (Dialog)* option, shown to the right, in which the extent of a box is specified and the mesh within the box is refined using the principles exemplified above. There is a tutorial dedicated to this feature.

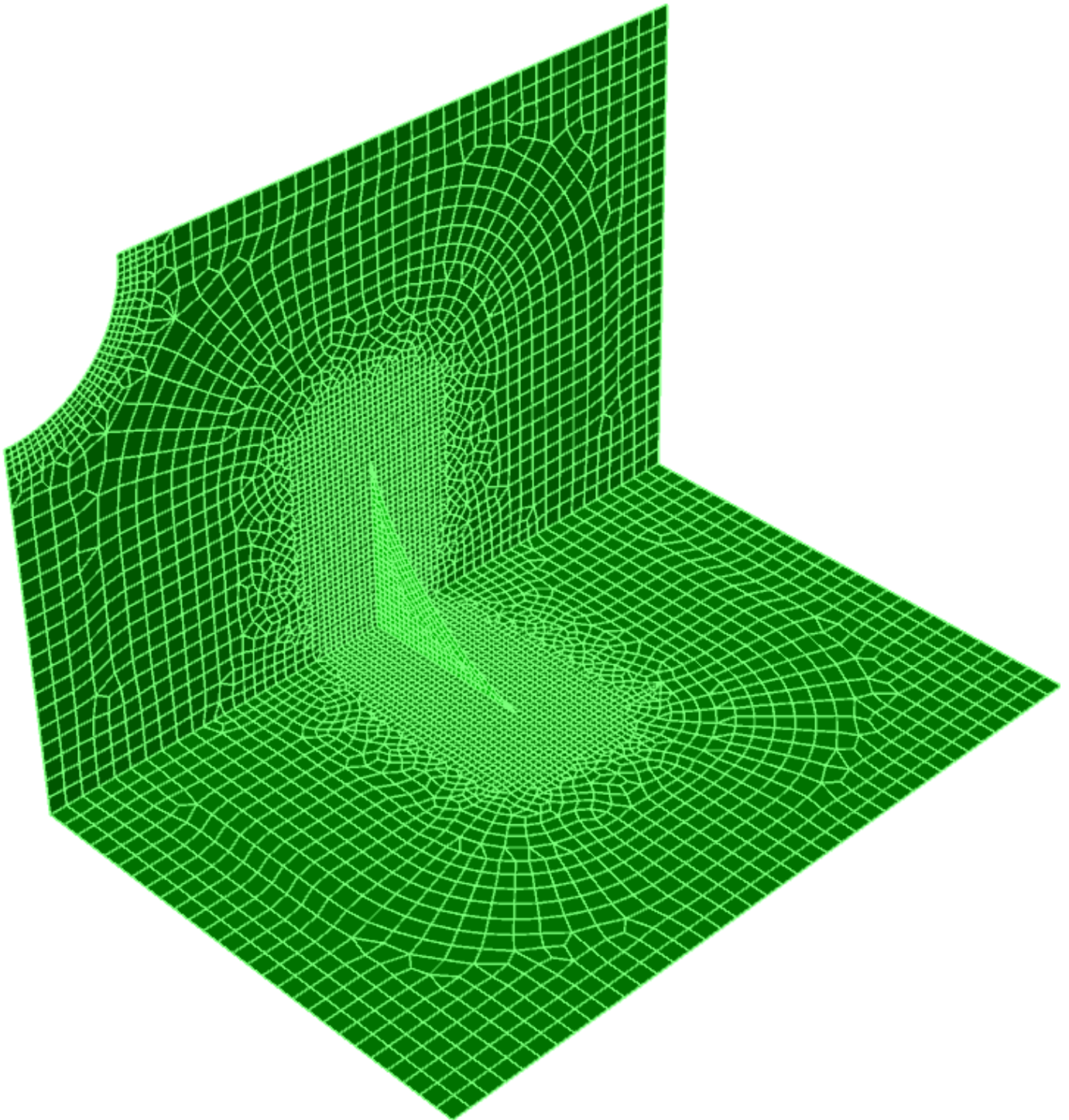


9 CREATE MESH FOR THE REMAINING PART

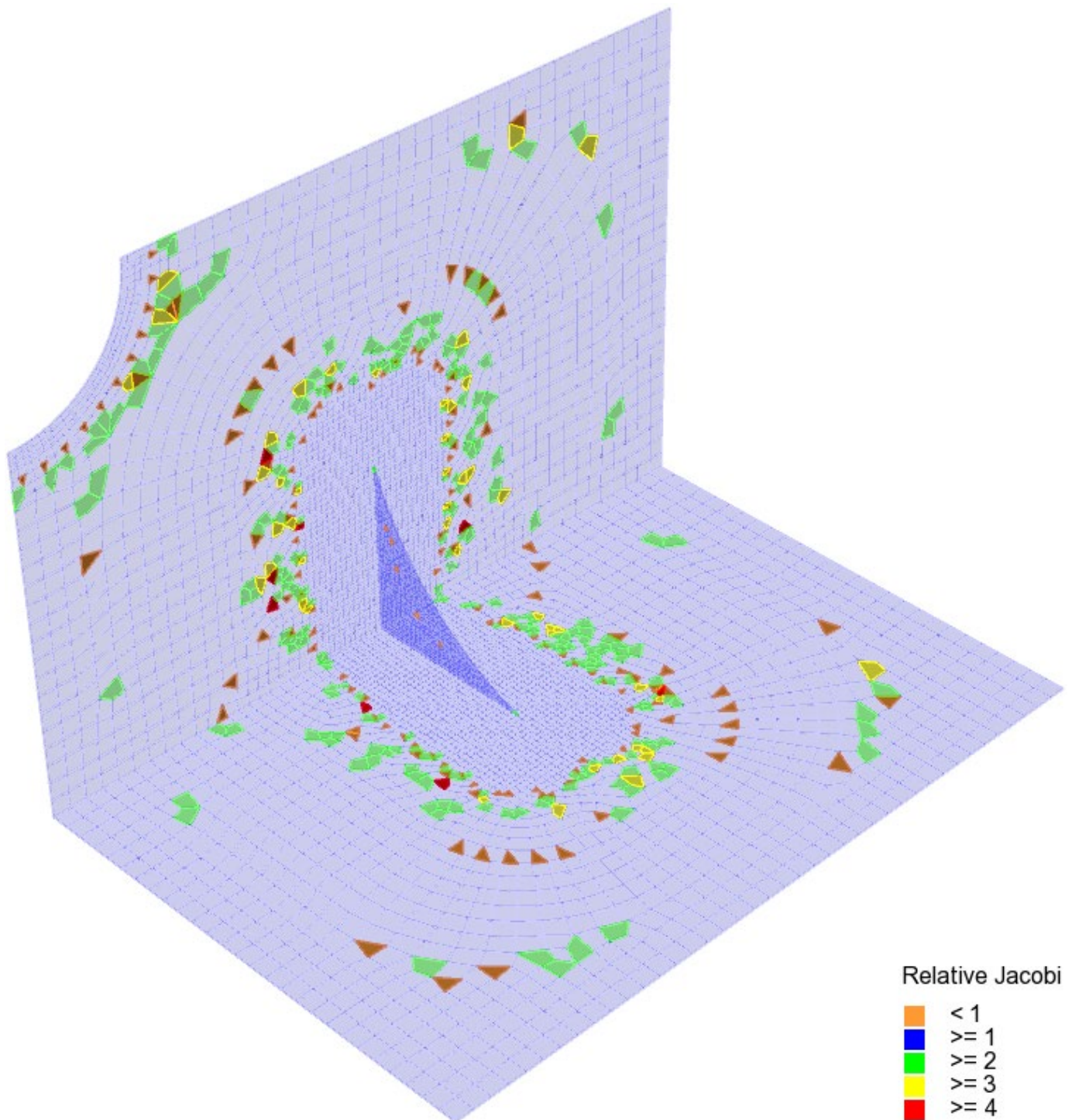
- The meshes for the sets 'fine' and 'coarse' are finalised.
- We will now create a mesh for the rest of the model.
 - Use Alt+M to open the *Activity Monitor* for the existing analysis activity.
 - Edit the meshing activity and in the *Mesh activity* dialog select *Subset <None>*.
 - Thereafter, click *Start* in the *Activity Monitor* to create a mesh.



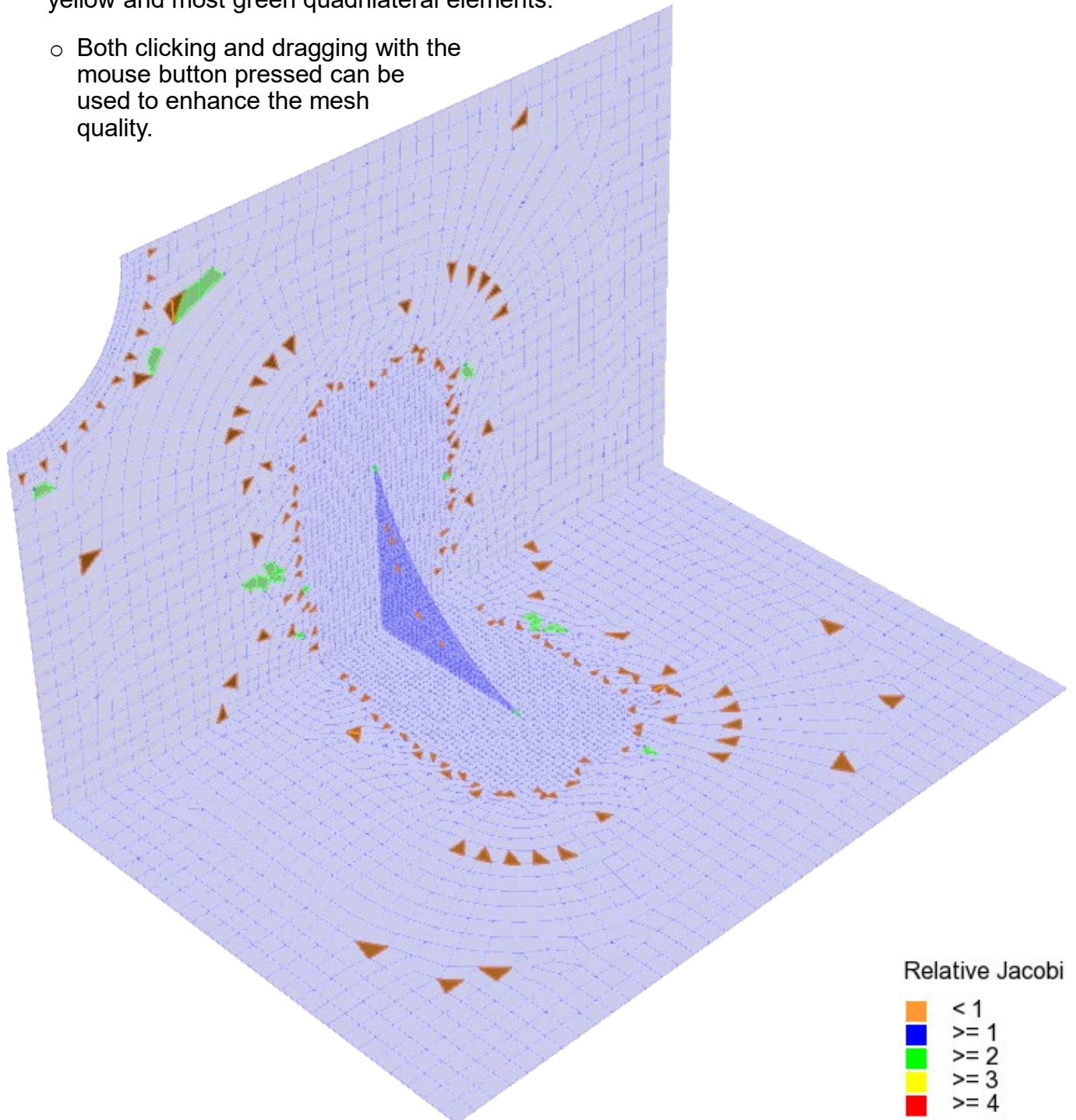
- Using Alt+A and pressing the function key F5 displays the current mesh for the whole model.
- All refinements of the above actions are maintained and a mesh with a gradual refinement towards the set 'fine' has been created for the set 'medium'.



- Selecting the whole model and colour coding the Jacobian determinant as before reveals some yellow (determinant > 3) and green (determinant > 2) quadrilateral elements you might want to fix by the *Enhance Quality* feature.
 - Note that the orange triangles cannot be removed this way.



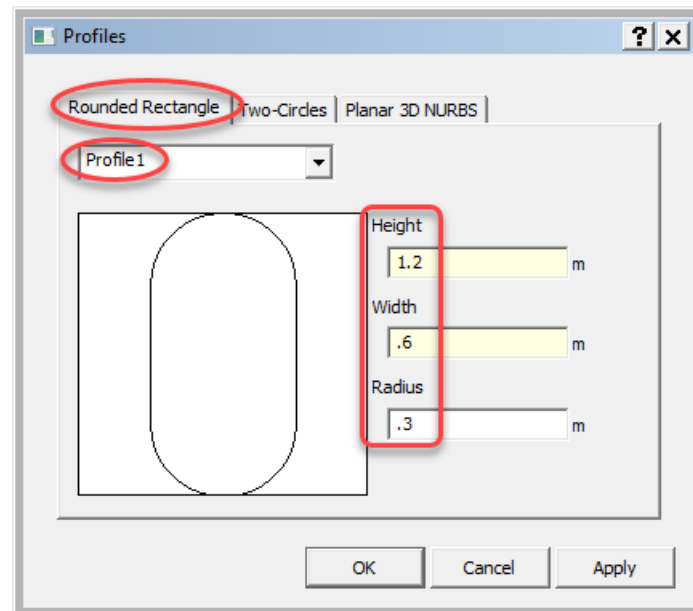
- The below display shows that the *Enhance Quality* feature has been able to remove all yellow and most green quadrilateral elements.
- Both clicking and dragging with the mouse button pressed can be used to enhance the mesh quality.



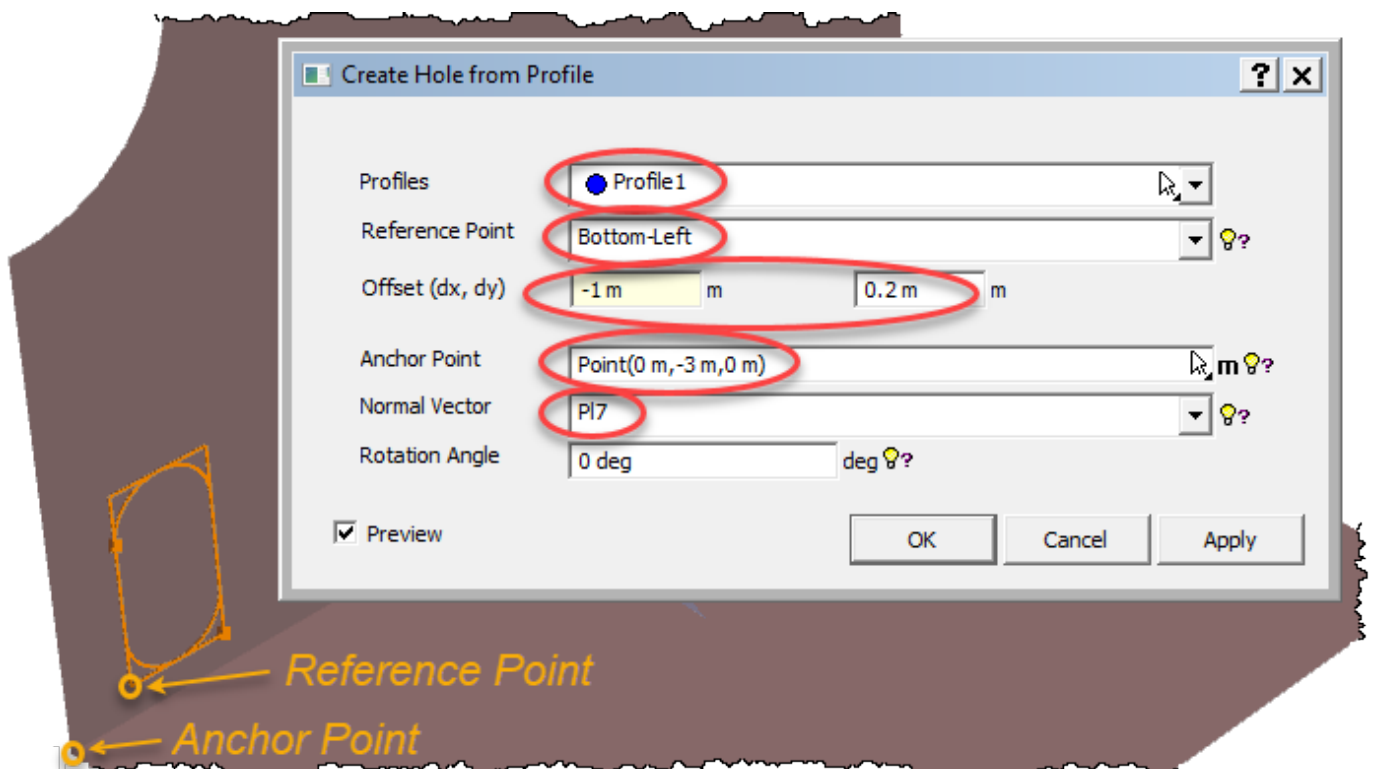
- Note that all mesh editing operations done above are performed on an existing mesh. They are, as for other operations in GeniE, logged. This means that reading the logged commands into a new workspace will redo also the mesh editing operations.
- But, recreating the mesh for a set containing the mesh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

10 ADD A HOLE CONCEPT AND CREATE A MESH AROUND THE HOLE

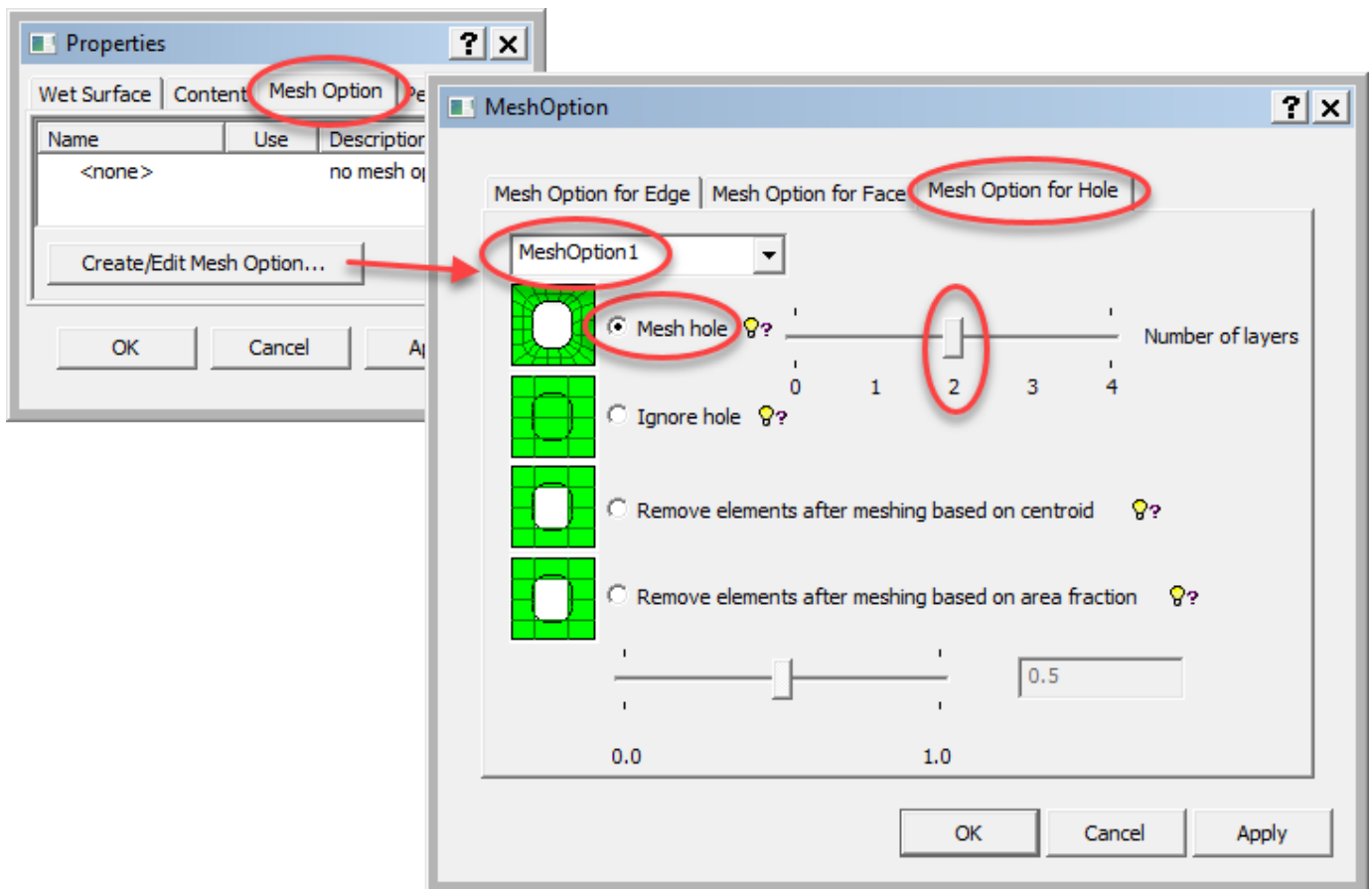
- A hole in a surface may be modelled using a hole concept. Introducing a hole concept does not remove the surface where the hole is positioned. Rather the surface 'inside the hole' is neglected when creating a mesh, computing loads, etc. Deleting the hole concept restores the surface without hole.
- Use *Guiding Geometry | Profiles | Profile Dialog* to create a profile as shown below.



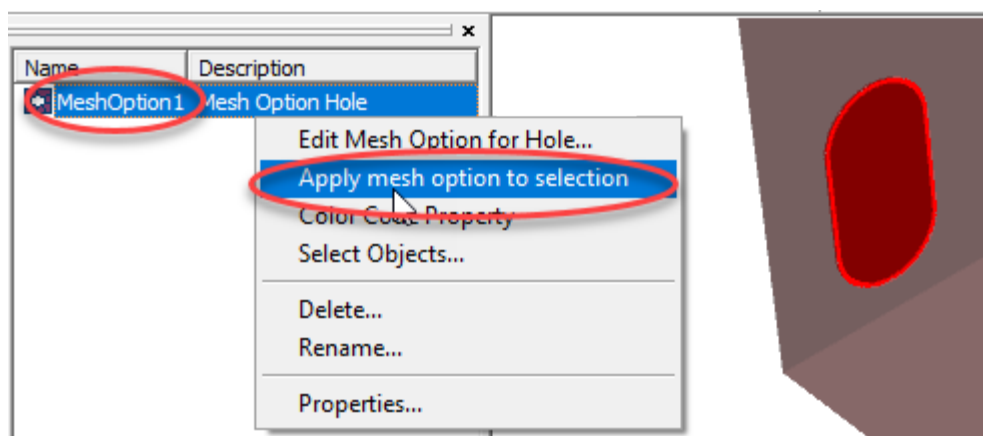
- Use *Structure | Features | Hole from Profile* to add a hole as shown below. The bottom left corner of the profile (*Reference Point*) is *Offset* (dx and dy) from an *Anchor Point* and with a *Normal Vector* taken from the selected plate. PI7 is the vertical plate.



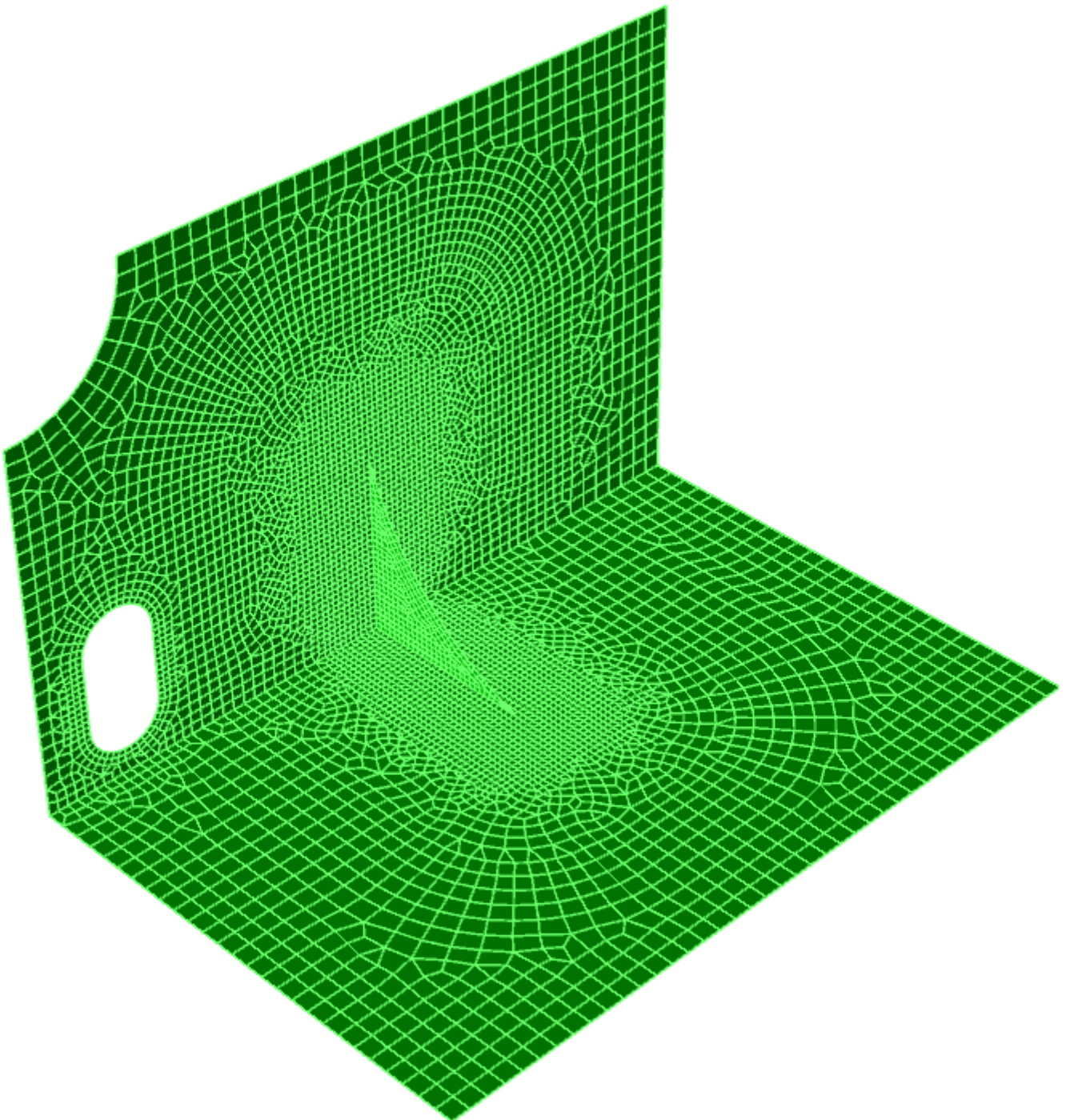
- Notice that the hole is found in the *Structure | Holes* folder in the browser. Deleting the hole concept removes the hole. If you try this, use undo (Ctrl+Z) to restore the hole.
- Use *Edit | Properties* and go to the *Mesh Option* tab in the *Properties* dialog. Click *Create/Edit Mesh Option...* and go to the *Mesh Option for Hole* tab to determine how to mesh the hole. Select *Mesh hole* with *Number of layers* of regular elements set to 2.
 - Notice the simplified ways of meshing the hole (*Remove elements ...*) in addition to the detailed *Mesh hole* option.



- Select the hole concept and assign the mesh option to the hole.



- Also, assign a fine mesh density to the hole concept, e.g. mesh050.
- Use Alt+M to open the *Activity Monitor* and click *Start* to create a new mesh. The new mesh is shown below.
 - Note that, as warned earlier in this tutorial, the mesh editing operations done in the upper left corner for the circular concavity are lost since the set named coarse has been remeshed.
 - This means that in this case the mesh edits should have been done after introducing the hole.





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.