



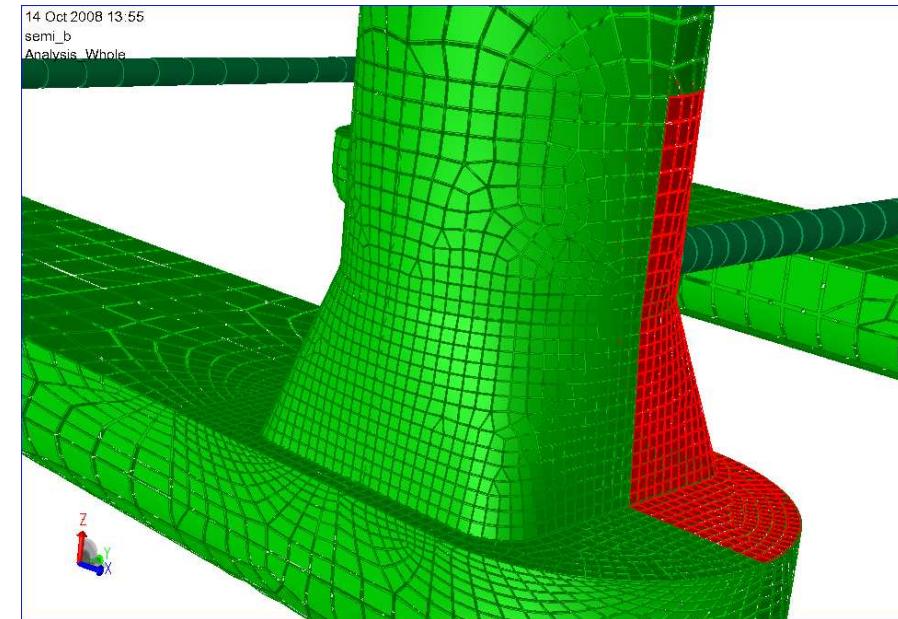
DNV Software - Sesam™

GeniE v5.1
Mesh guidance

Revision date 25 February 2010

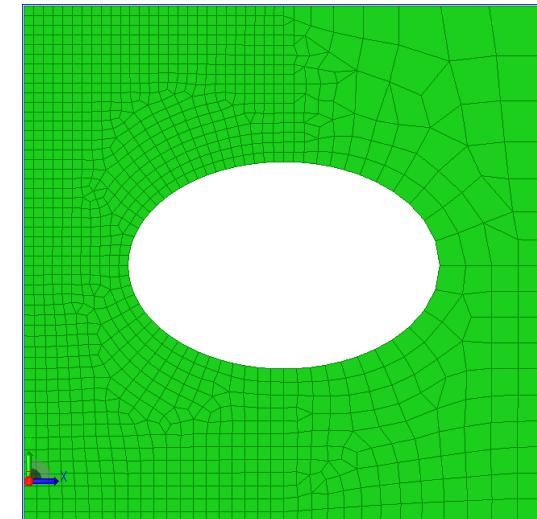
Most common used mesh controls

- The basic mesh controls can be applied to the entire model or vary from part to part
 - 1. Type of elements
 - Shell vs. membrane, first or second order elements
 - 2. Mesh density
 - Decide the size of the FE mesh
 - 3. Mesh gradients
 - Controlling the transition zone from fine to coarse mesh
 - 4. Mesh algorithms
 - Quad vs. paver meshing
 - 5. Feature edges
 - Insert mesh lines manually
 - 6. Prioritized meshing
 - The sequence to generate the mesh



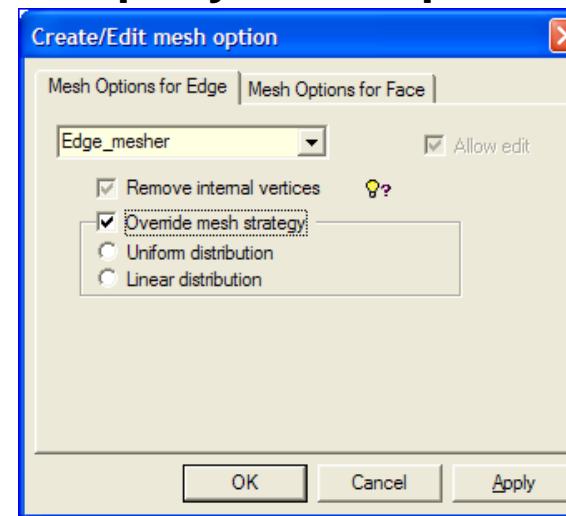
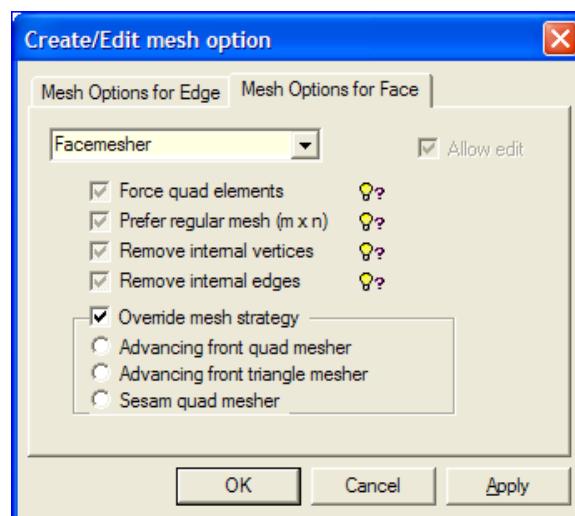
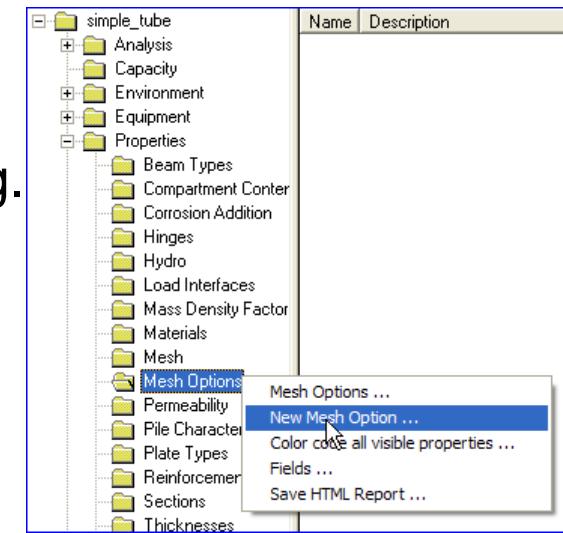
Access to mesh control

- Meshing rules
 - Global mesh control parameters
 - Activated from the *Edit/Rules/Meshing*
- Type of elements
 - Global and local settings
 - Accessed from the **Property Plate Type** (and Meshing Rules)
- Mesh density properties
 - Global default values as well as local mesh densities
 - Accessed from the **Property Mesh browser**
- Mesh options
 - Local options overriding the setting as defined by the meshing rules
 - Accessed from the **Property Mesh Options browser**
- Prioritized meshing
 - Local option specifying the order of meshing
 - Accessed from the **Utilities browser**
- Feature edges
 - Local options to ensure mesh line at desired locations
 - Accessed from **Structure Features browser**



Global vs. local mesh settings

- Global mesh settings apply to the entire model
- Local mesh settings apply to parts of the model, e.g.
 - Mesh densities
 - Feature edges
 - Mesh locking
 - Prioritized meshing
 - Mesh options overriding global settings
 - Accessed from the **Property Mesh Options browser**

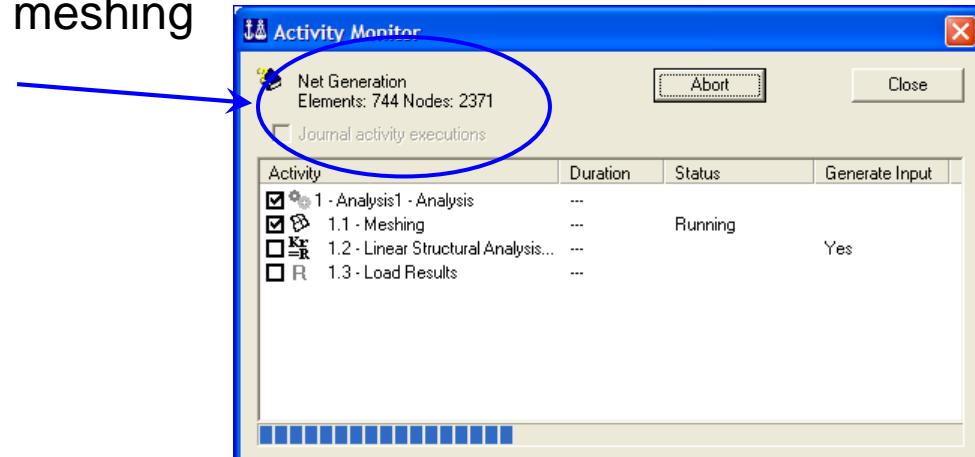


There may be several
mesh options per model

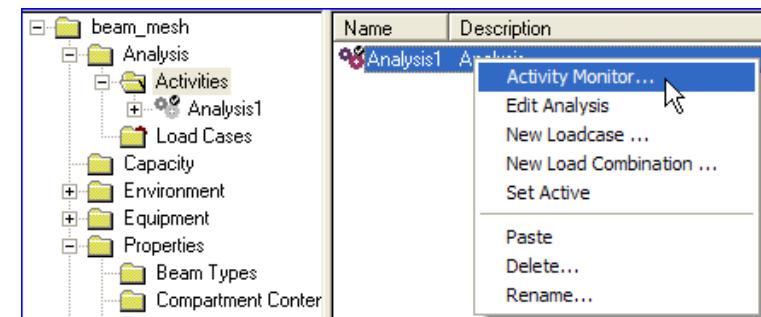
There can only be one
global setting per model

Create a finite element mesh

- The mesh can be created from
 - *Tools|Analysis|Create Mesh* or short command **Alt+M**
 - *Tools|Analysis|Activity Monitor* or short command **Alt+D**
 - Deselect other activities than meshing
 - Will show status on meshing

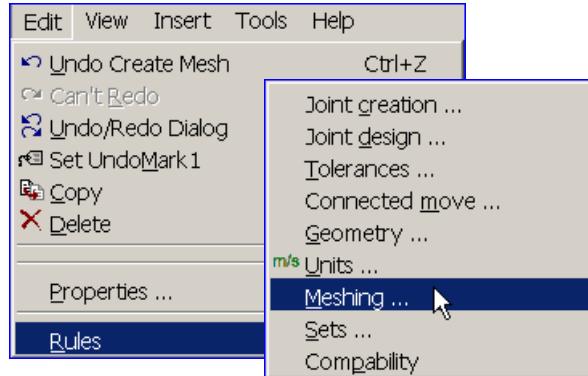


- From browser Analysis Activities
 - Create activity and right click to start the Activity Monitor

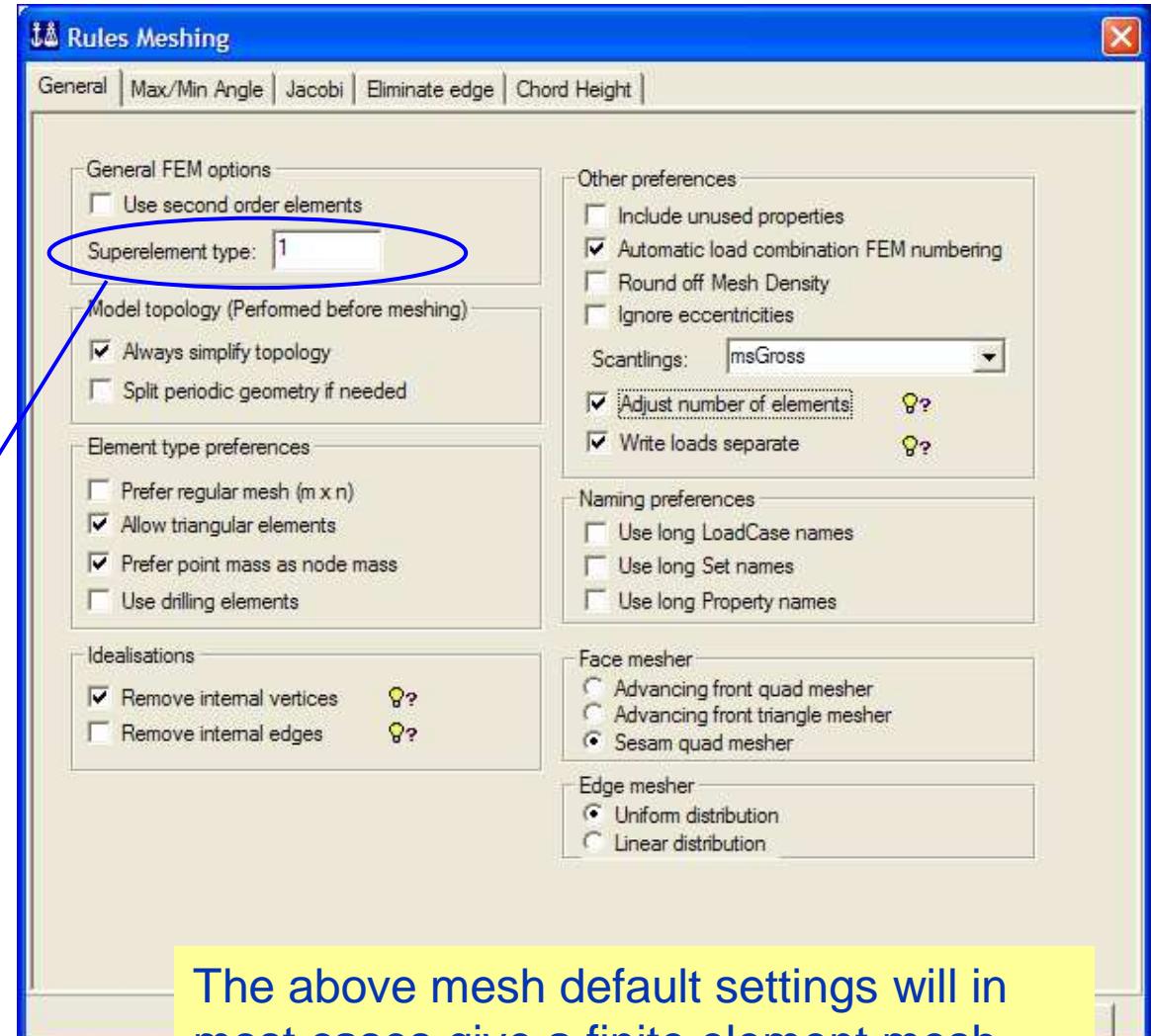


Activating global mesh control parameters

■ Edit|Rules|Meshing

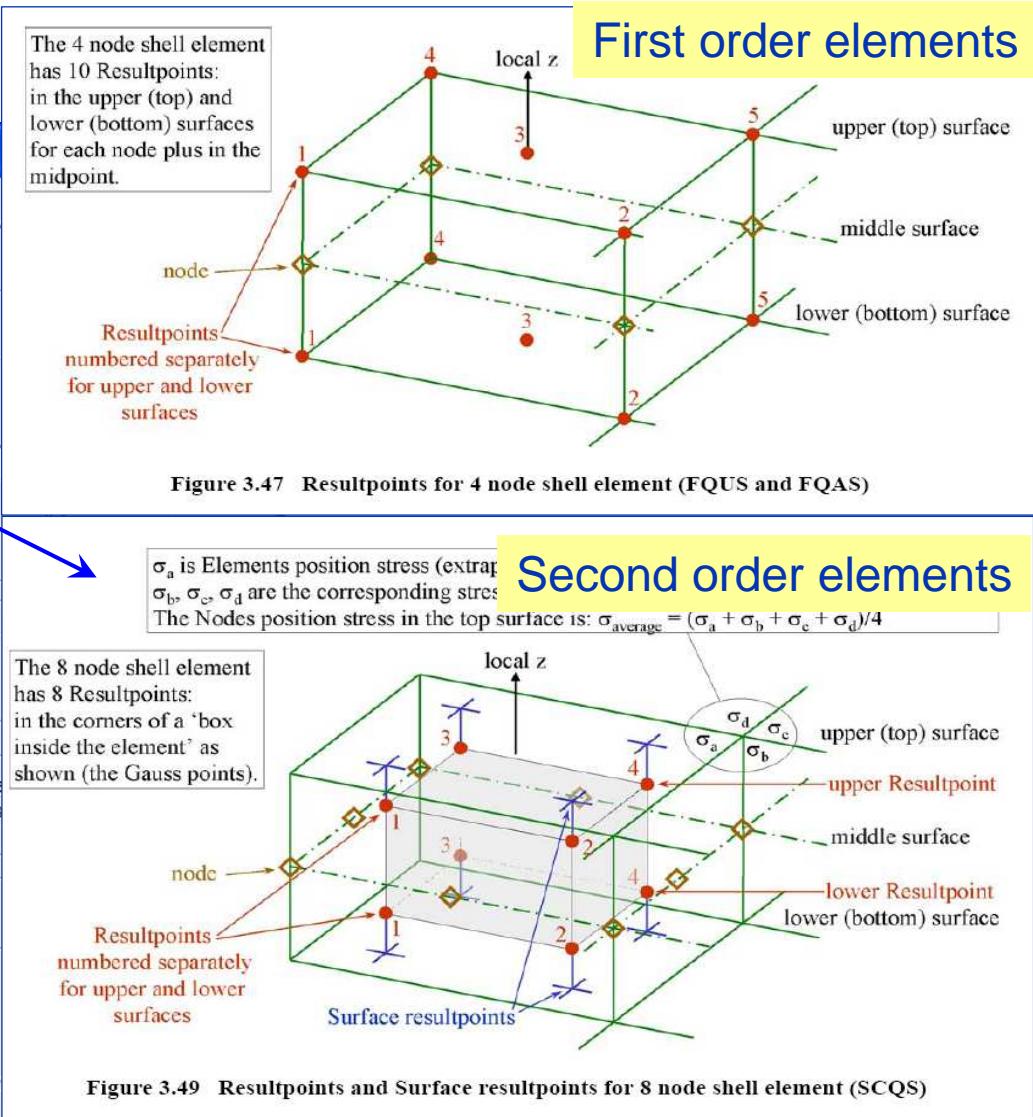
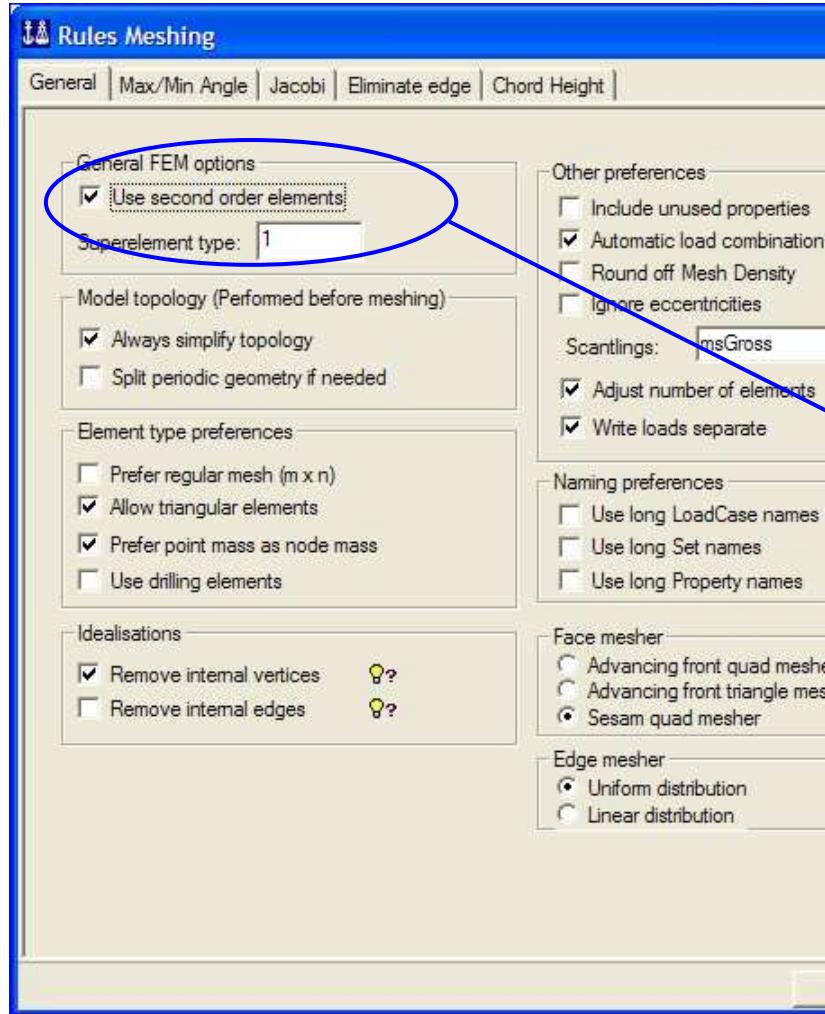


- The superelement type is used to define super-element numbers for use in external analysis
 - e.g. Presel, Sestra, HydroD
 - Tn.FEM



Global and general mesh settings

■ First or second order elements



Global and general mesh settings

- Element types
 - Those supported by GeniE
- Not possible to mix 1st and 2nd order elements
- Possible to mix thin shell and membrane elements

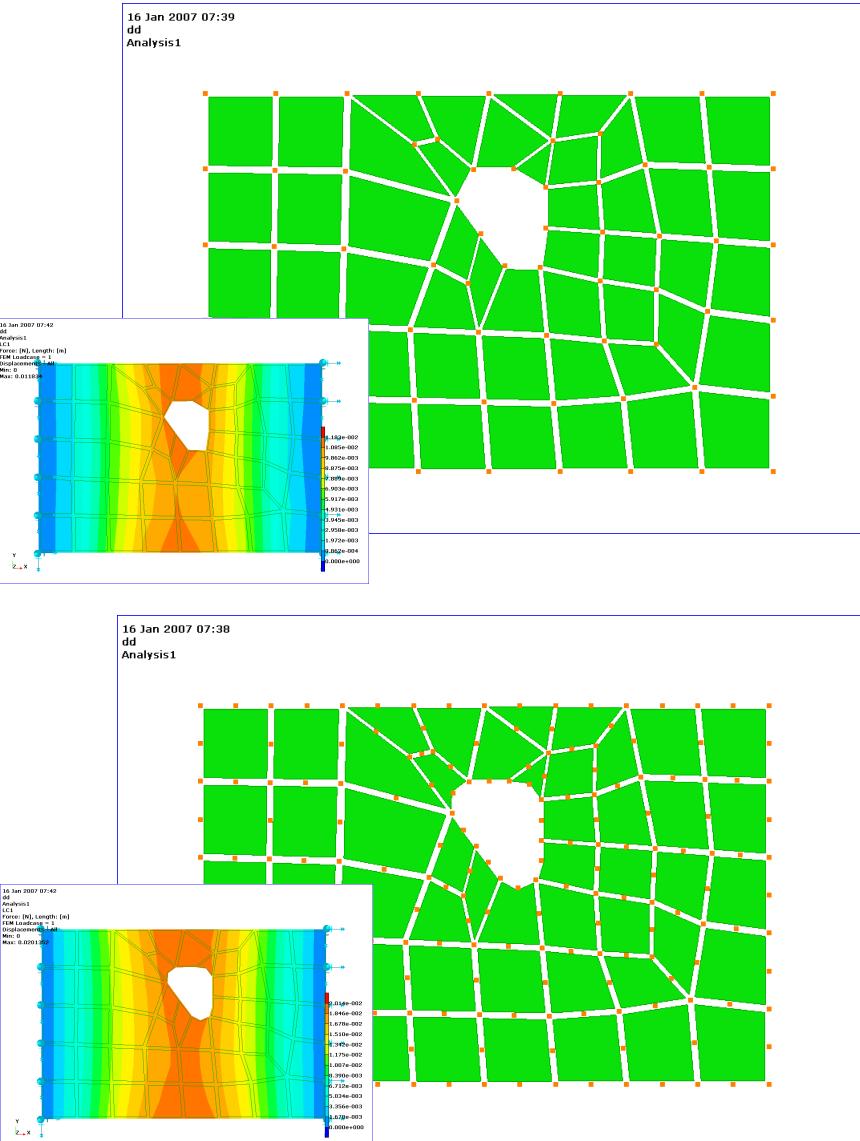
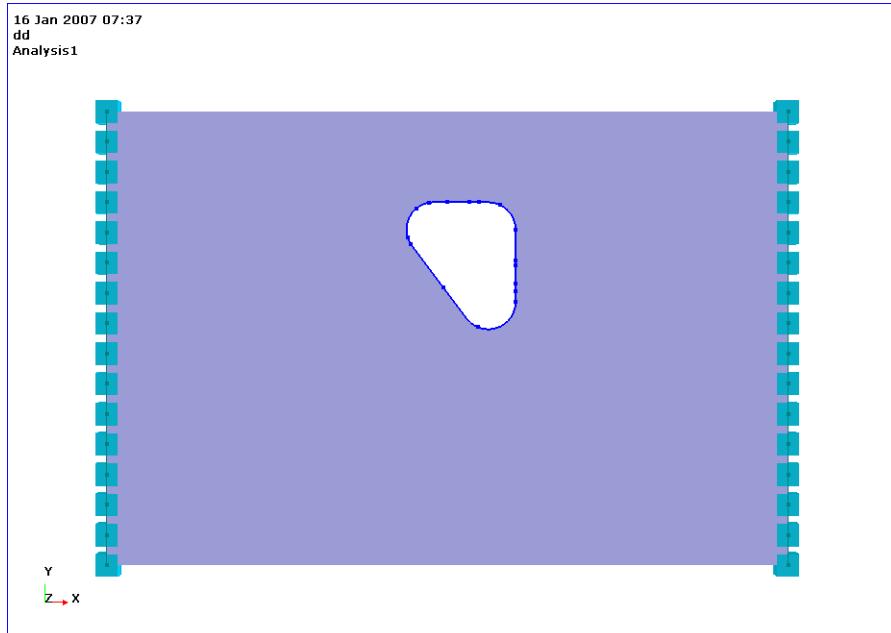
The following finite element mesh types are generated and exported to the FEM file. More details about each element may be found in Sestra User Manual.

Name	Type	Order	Comments
2-node beam element	BEAS	1 st	
3-node beam element	BTSS	2 nd	Cannot be used in framecode checking
Quadrilateral flat thin shell element	FTRS	1 st	
Triangular flat thin shell element	FQUS	1 st	Inserted when adjusting mesh rules to split elements
Quadrilateral sub parametric curved thick shell element	SCQS	2 nd	
Triangular sub parametric curved thick shell element	SCTS	2 nd	Inserted when adjusting mesh rules to split elements
Quadrilateral flat thin shell with drilling dof	FQAS	1 st	Includes the rotational dof around the axis perpendicular to the membrane in the membrane formulation
Triangular flat thin shell with drilling dof	FTAS	1 st	- o -
Non-structural 2 node beam element	BEAS	1 st	Special variant of BEAS with no contribution of the structural stiffness
Truss element	TESS	1 st	Element type with no bending stiffness
Spring to ground	GSPR	1 st & 2 nd	Includes the 6x6 matrix
Shim element	GLSH	1 st & 2nd	Special variant of the 2 node spring element with equal stiffness in two translation directions. No stiffness in other directions.
One node mass element	GMAS	1 st & 2nd	May be eccentric if connected to a finite element node with 6 dof.

Within the same model it is not possible to have both 1st and 2nd order element types.

Global and general mesh settings

- First order shell elements
 - 3/4 nodes per element
 - Sensitive to skewness
- Second order shell elements
 - 6/8 nodes per element
 - Larger FE models, but more accurate



Global and general mesh settings

■ Membrane element

- Define a plate membrane property and assign to the plate
- The mesh automatically becomes membrane elements (first or second order)
- No stiffness normal to plate and can not support normal pressure load

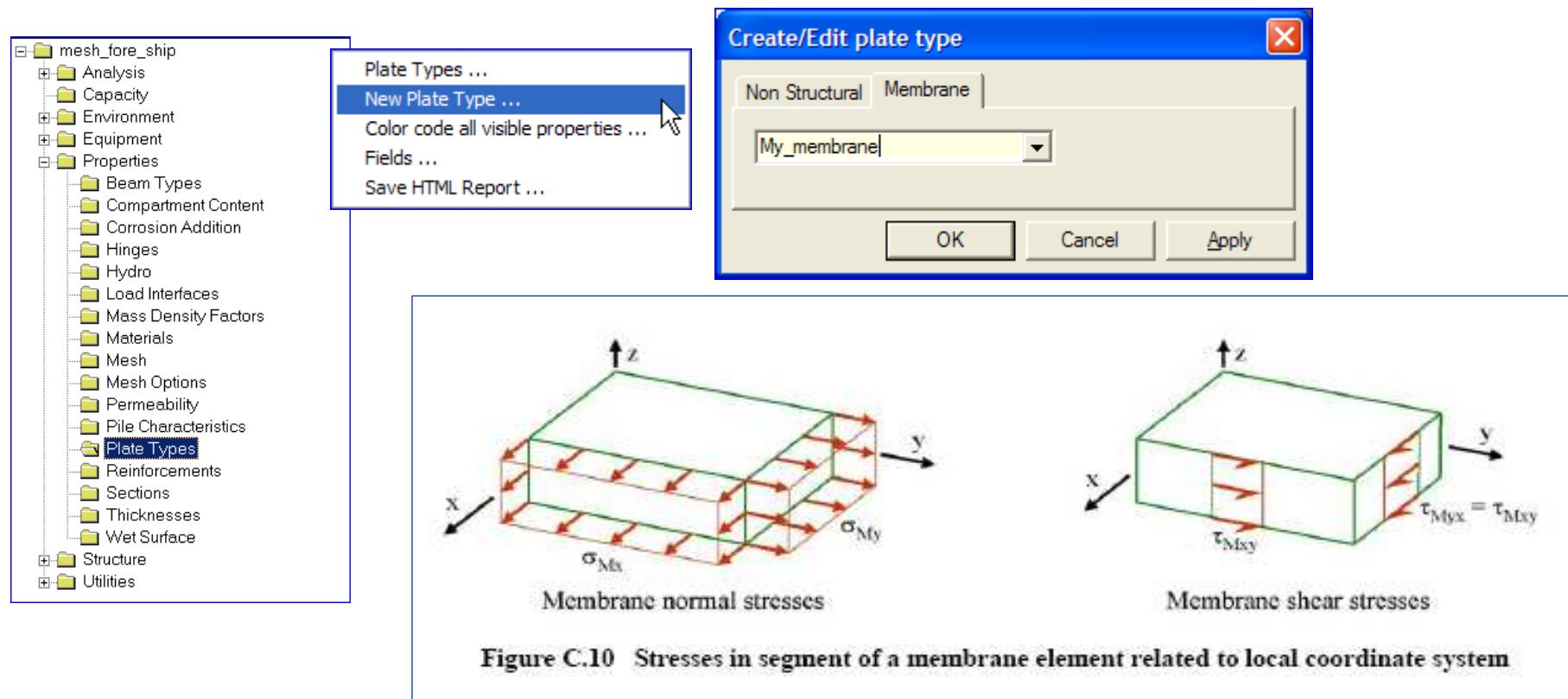
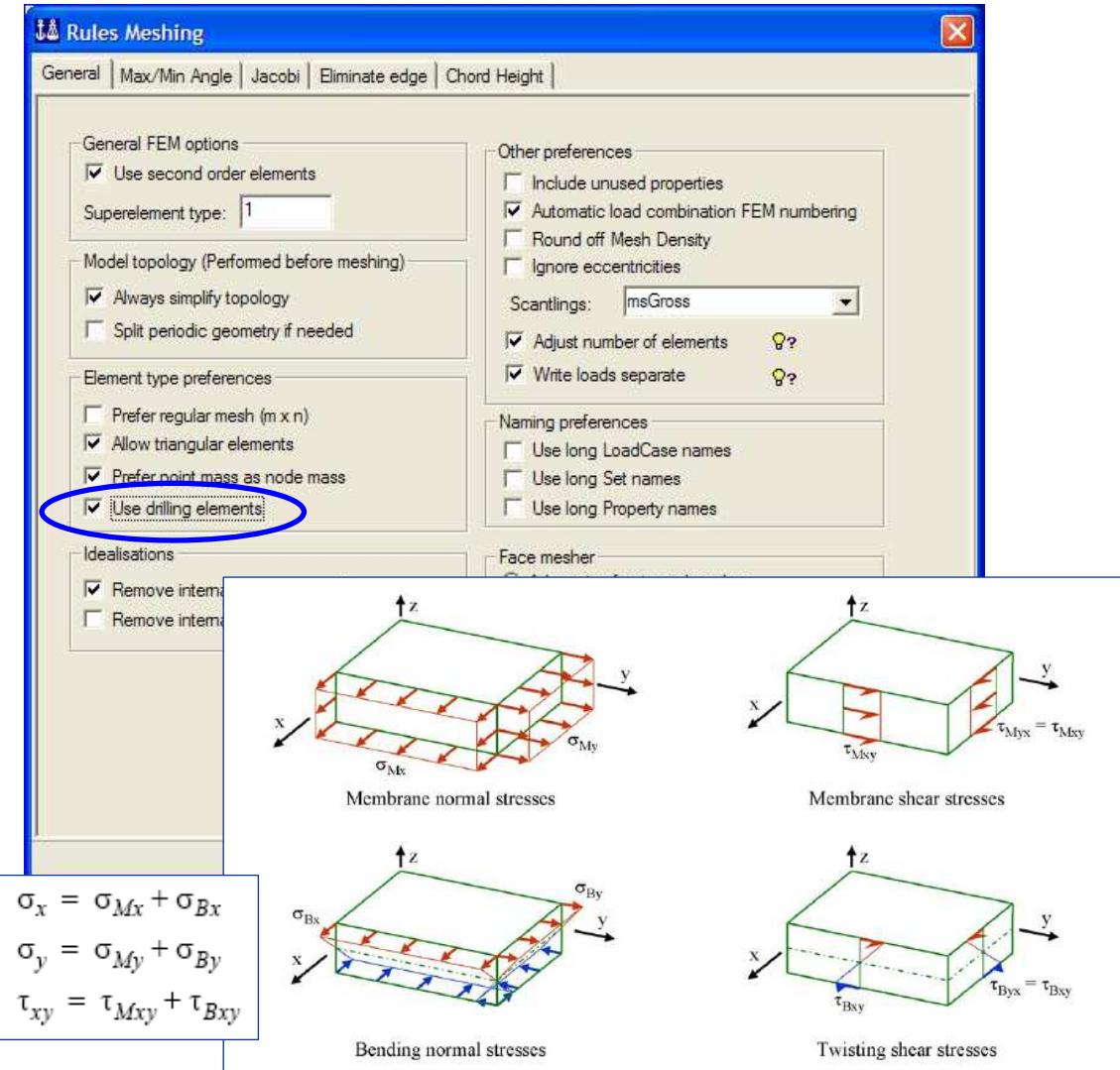


Figure C.10 Stresses in segment of a membrane element related to local coordinate system

Global and general mesh settings

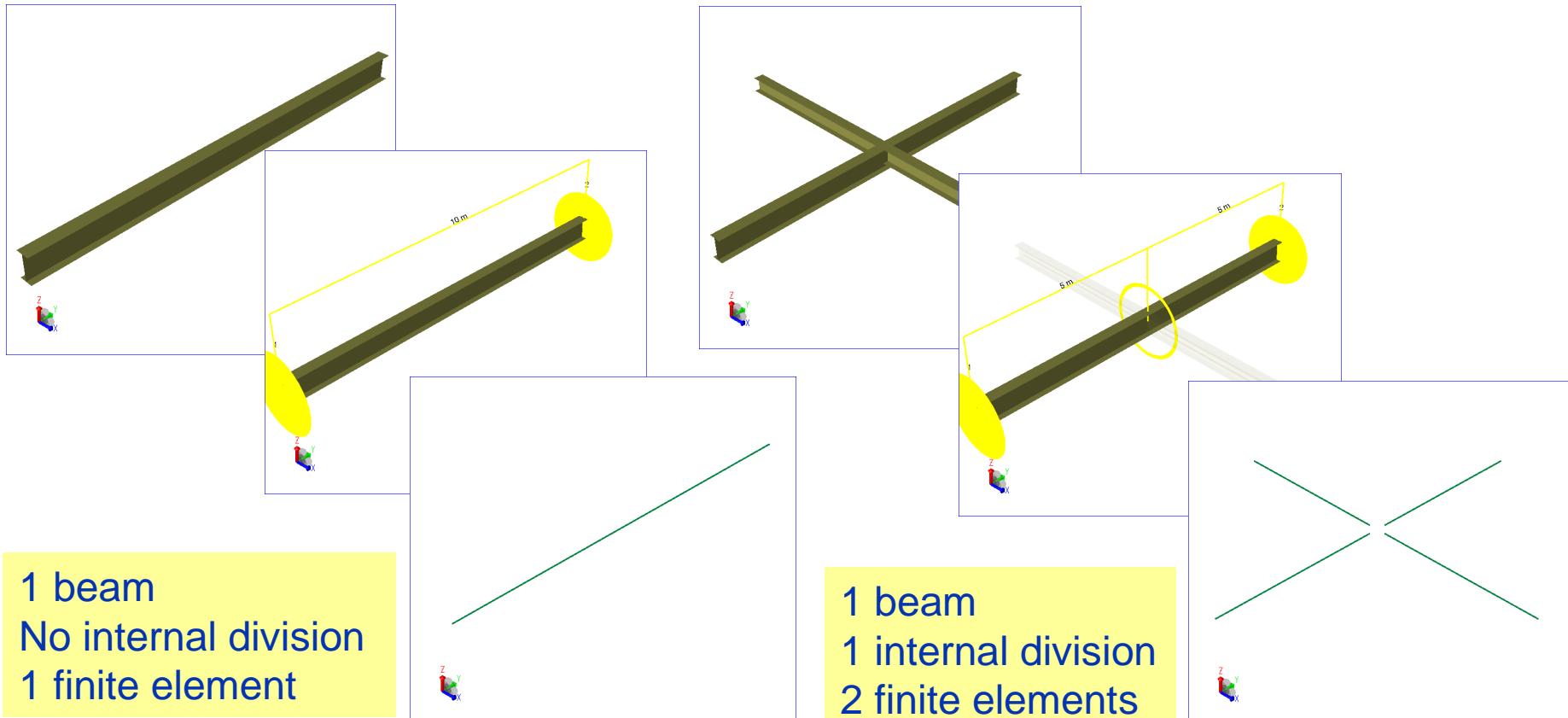
■ Drilling element

- Special variant of the thin shell element:
 - Independent membrane and plate bending parts
 - Utilizes the rotational degrees of freedom around the axis perpendicular to the membrane plane in the membrane formulation
 - The element is very stiff when membrane action dominates but is quite good for plate-bending



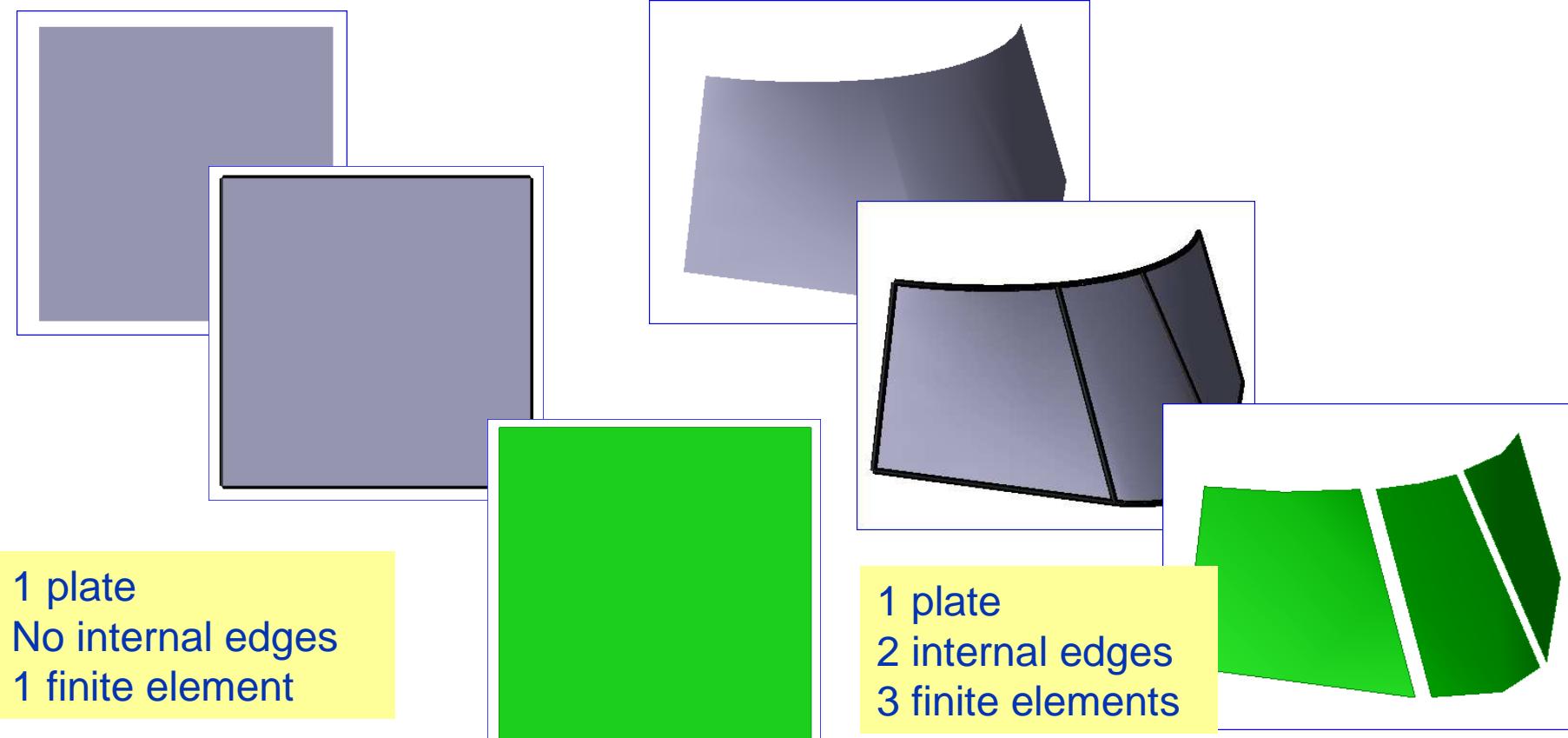
Default mesh for beams

- The default mesh size is one finite element per beam
 - If beam is split in parts (or segments), then one finite element per part (double click to see segments)



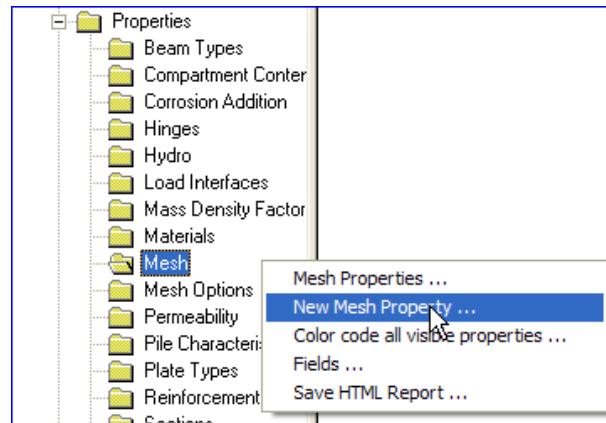
Default mesh for plates and shells

- The default mesh size is one finite element per surface
 - If surface split in several patches by internal edges, then one finite element per patch is the default (double click a surface to see the internal edges)



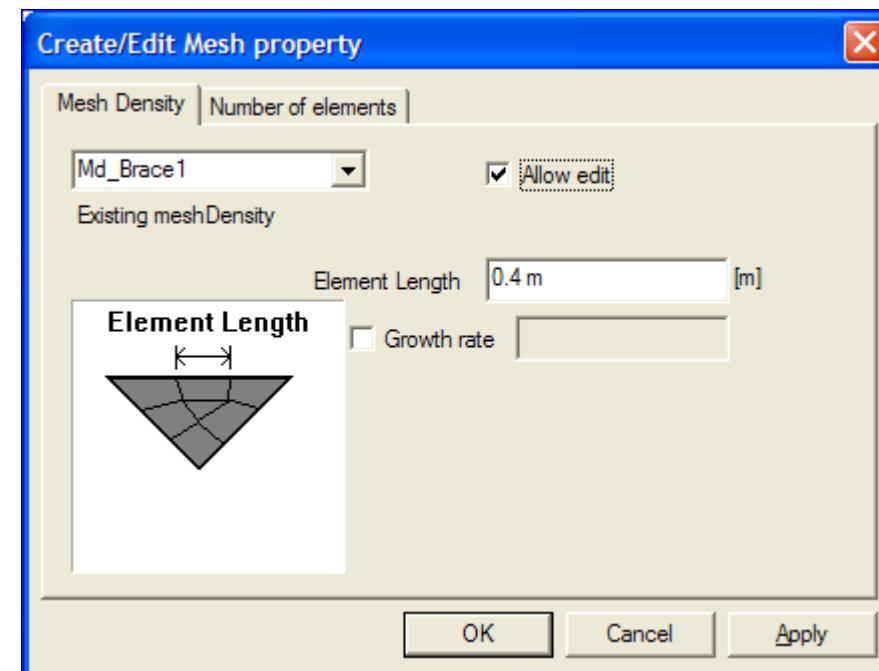
Define mesh sizes

- Mesh density located in browser **Properties Mesh**



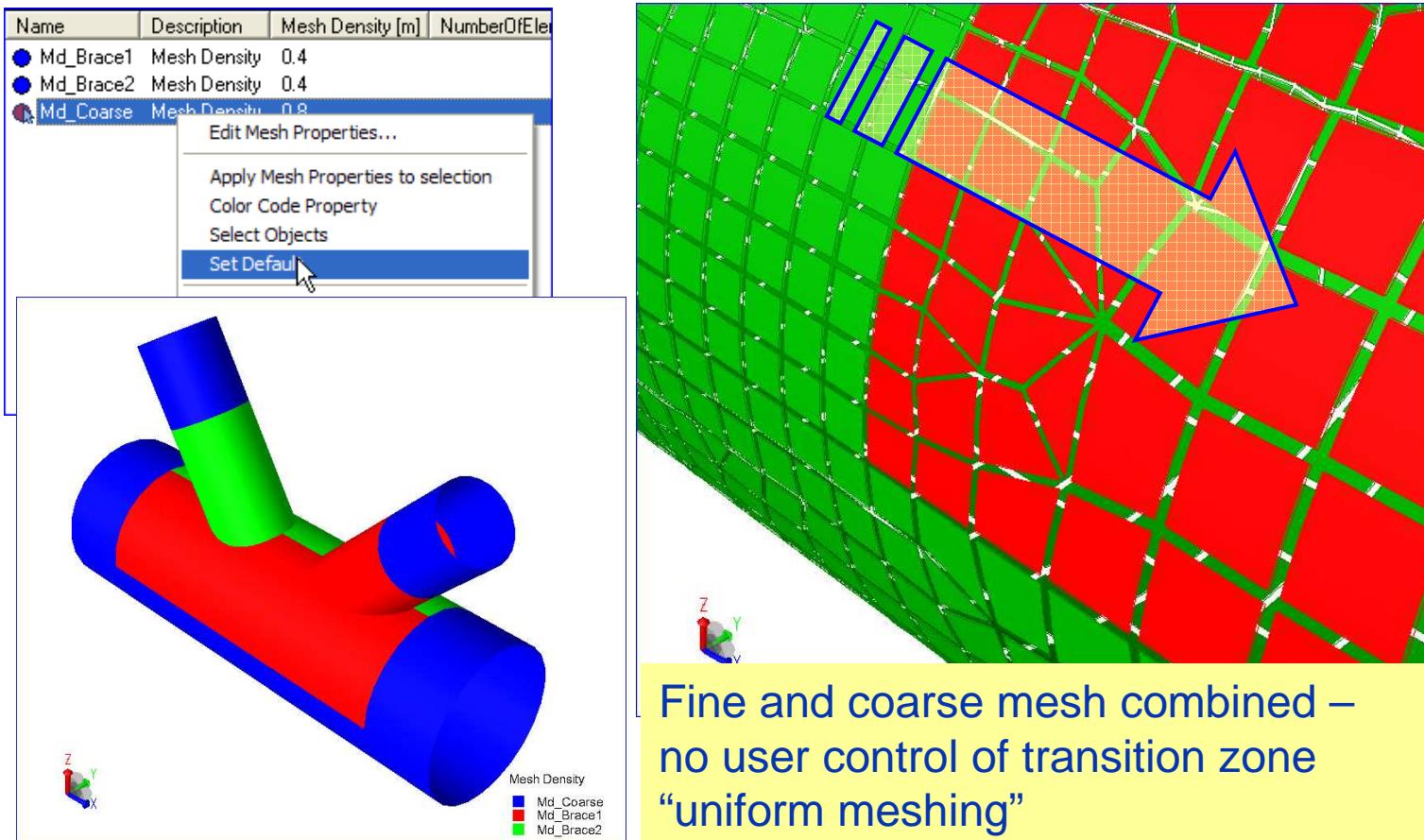
Name	Description	Mesh Density [m]
Md_Brace1	Mesh Density	0.4
Md_Brace2	Mesh Density	0.4
Md_Coarse	Mesh Density	0.8

Multiple mesh densities can be defined



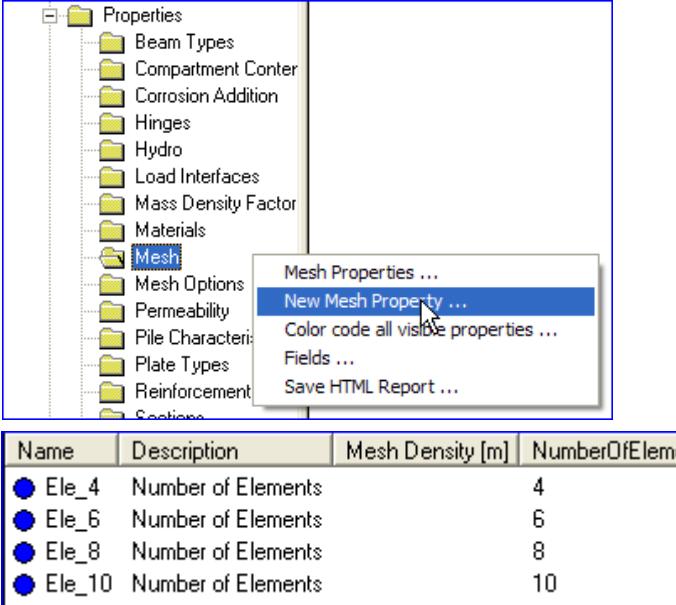
Assign mesh size to different parts

- The mesh properties can be assigned to different parts of model
 - Assigned to beams, surfaces and feature edges
 - A mesh density set to default is override by a mesh density assigned to a part

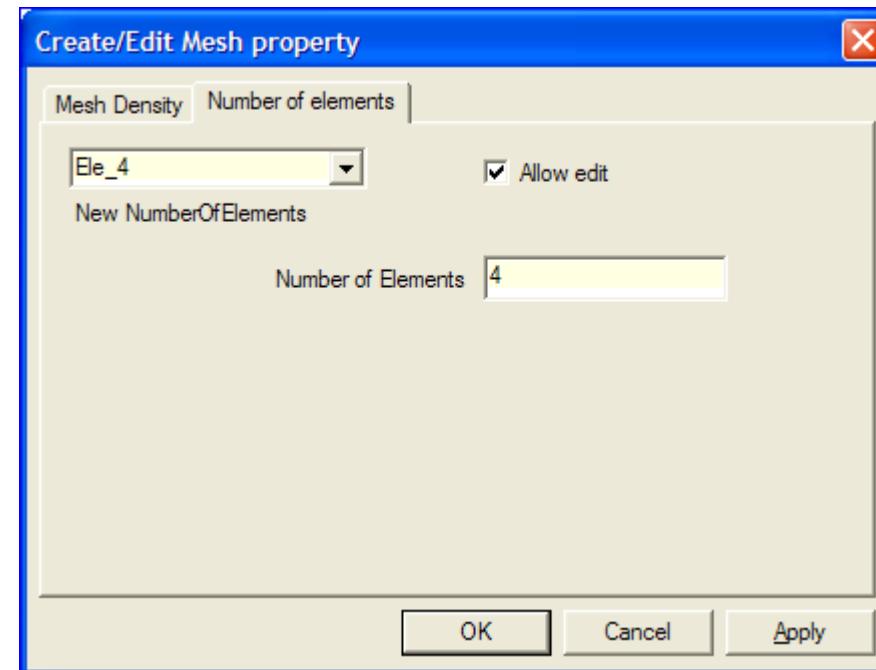


Define mesh sizes

- Number of elements located in browser **Properties Mesh**



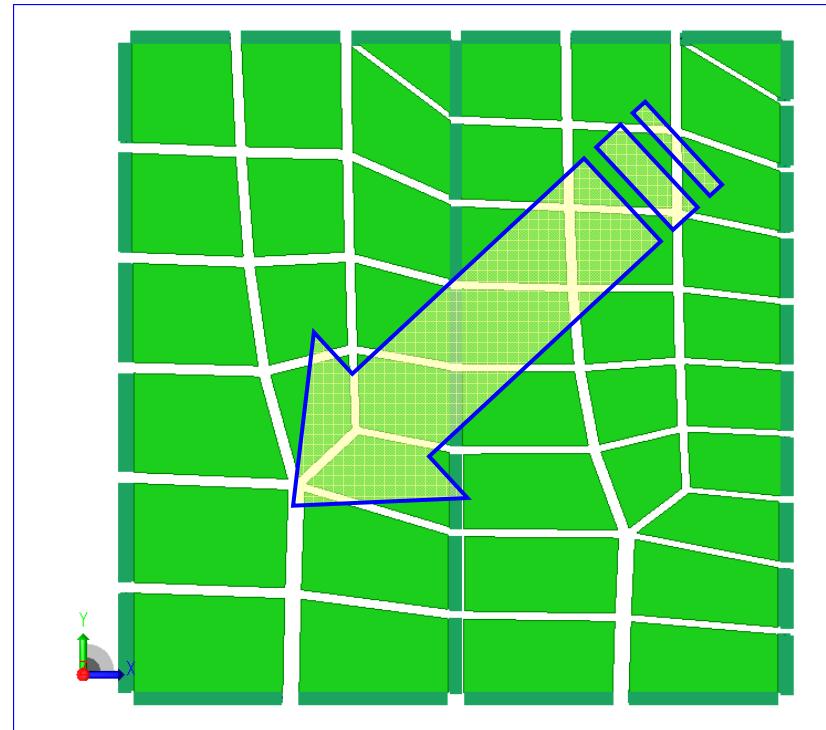
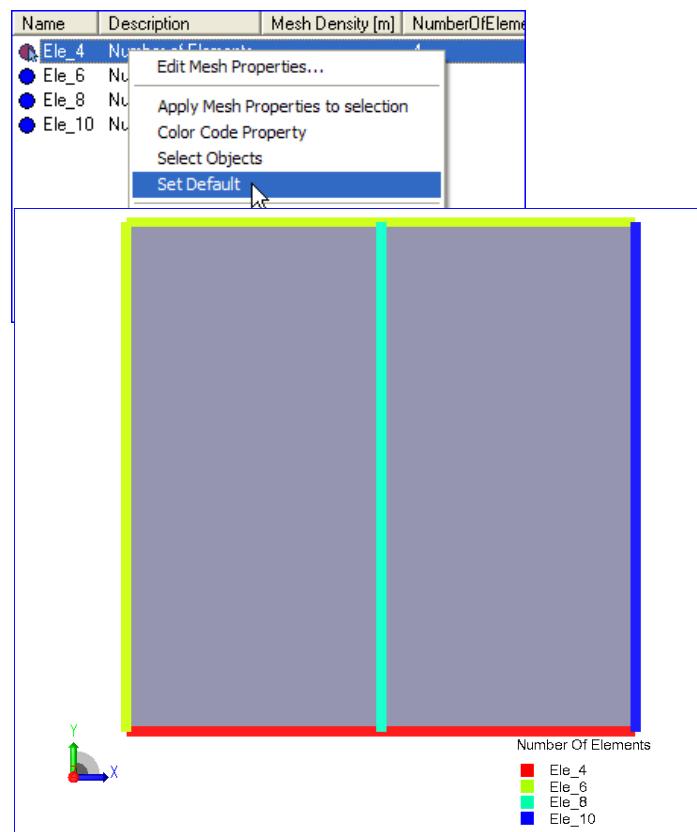
Name	Description	Mesh Density [m]	NumberOfElements
Ele_4	Number of Elements	4	
Ele_6	Number of Elements	6	
Ele_8	Number of Elements	8	
Ele_10	Number of Elements	10	



Multiple element lengths
can be defined

Assign element lengths to different parts

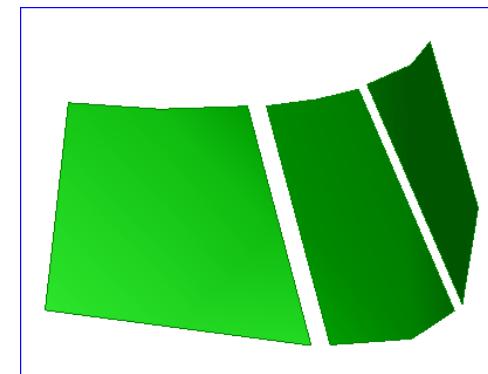
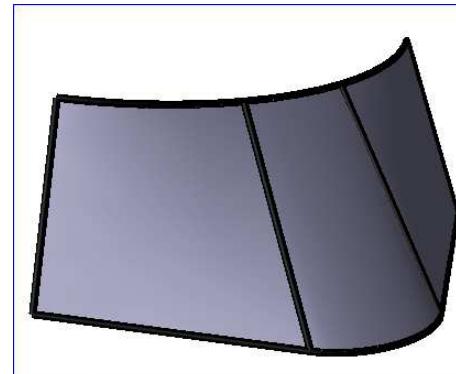
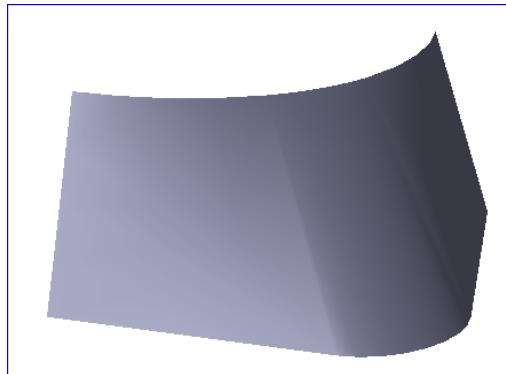
- The element length properties can be assigned to different parts of model
 - Assigned to beams and feature edges
 - An element length set to default is override by a mesh density assigned to a part



From many to few finite elements per beam

Mesh size - priorities

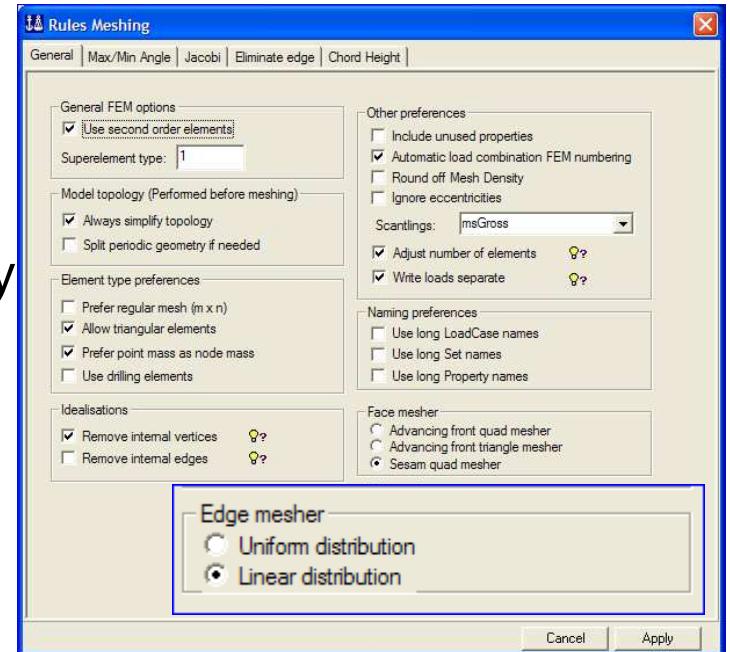
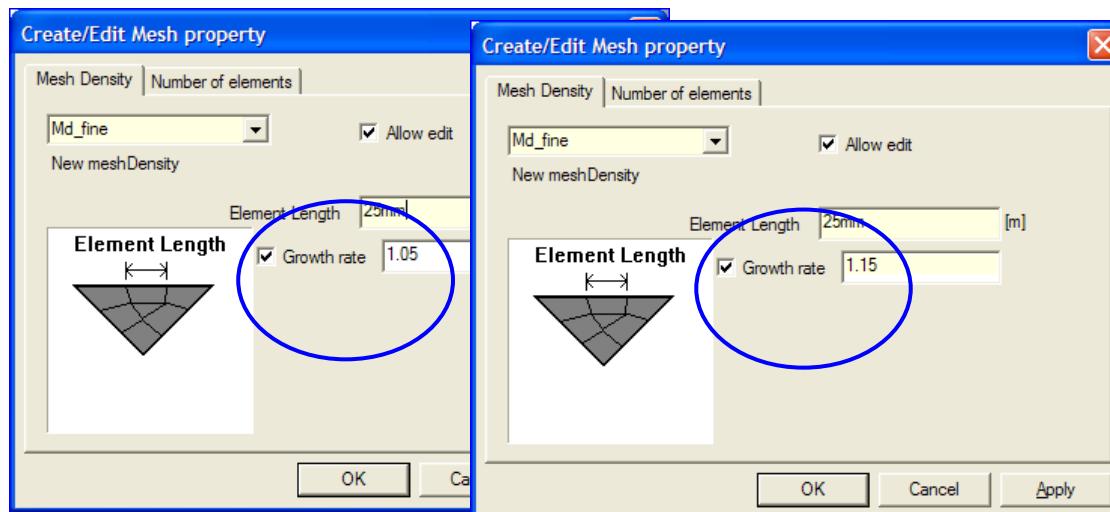
- GeniE uses the following priorities between mesh densities when creating the finite element mesh
 - 1. Number of elements along a line (beam or feature edge)
 - 2. Mesh density applied to a line (beam or feature edge) and a plate
 - 3. Global settings specified in the mesh rules
 - 4. If there are no settings, there will be 1 element per edge



No settings specified, hence settings from edges used to decide the mesh size, i.e. 1 element per edge in this case

Controlling the mesh transition zone

- Mesh transition zone = Length of zone from fine to coarse mesh
- To control the mesh transition zone
 - Activate Linear distribution on Edge mesher in the mesh rules
- Decide the growth rate for each mesh density to be part of a linear mesh distribution
 - The larger growth rate normally the longer transition zone

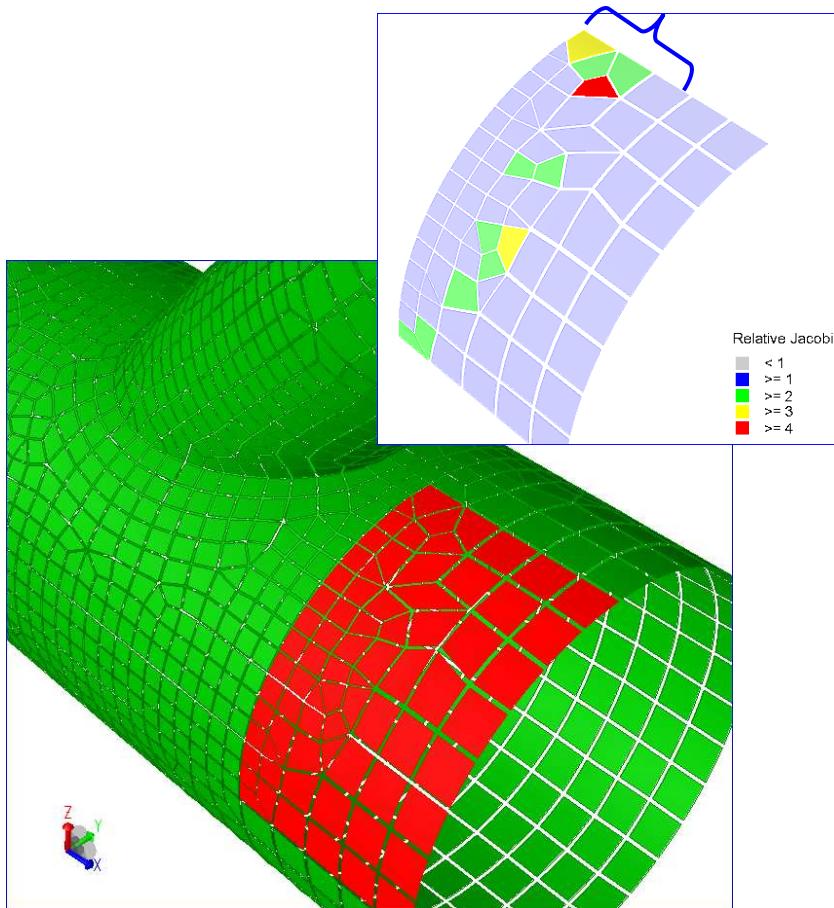


Linear mesh distribution applies to surfaces only

Controlling the mesh transition zone

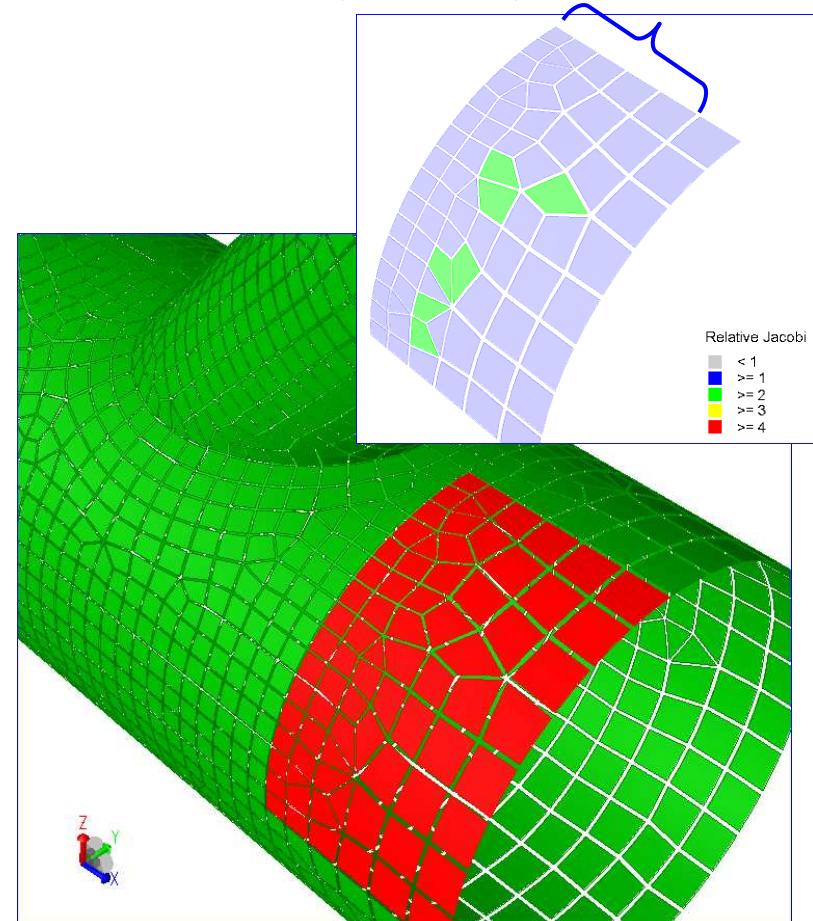
■ Uniform distribution

- The transition zone is as short as possible



■ Linear distribution

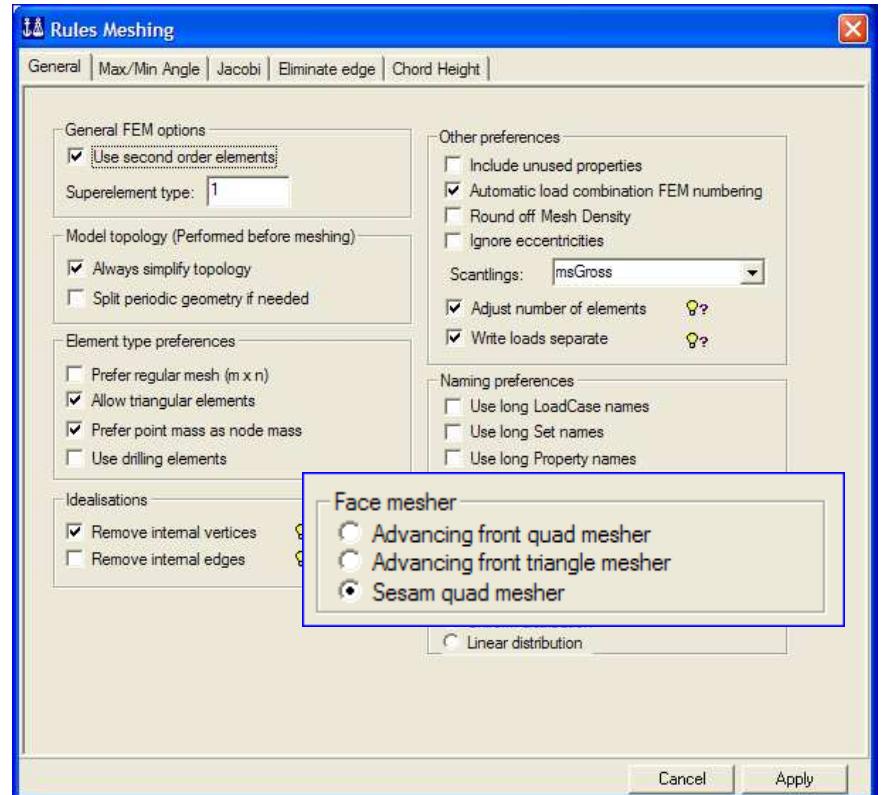
- The transition zone is prolonged and mesh quality normally better



Quad vs. paver meshing

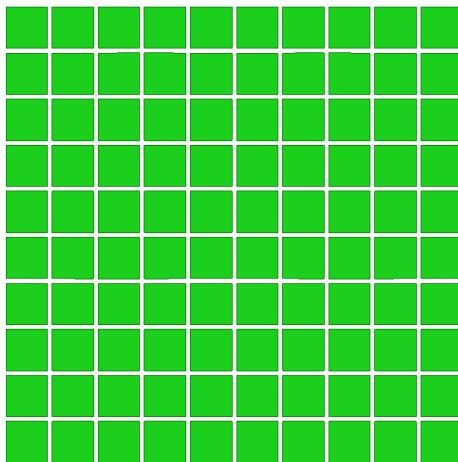
- Activating the different meshing algorithms from the mesh rules
- The default is the “Sesam quad mesher”
 - Fast and best for regular structures
- Options are “Advancing front quad mesher” and “Advancing front triangle mesher”
 - Best for structures with details like cut-outs and high degree of curvature

It is possible to have a mix of quad and paver meshing in same model

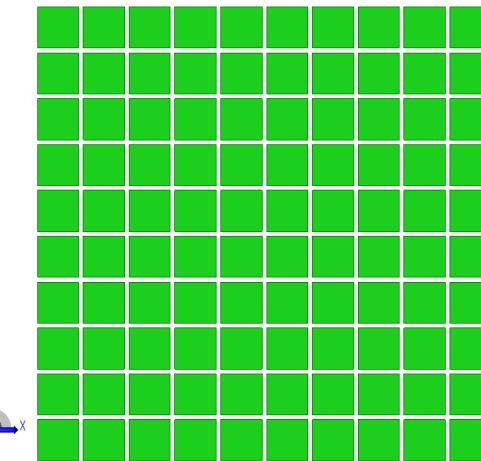


Quad vs. paver meshing

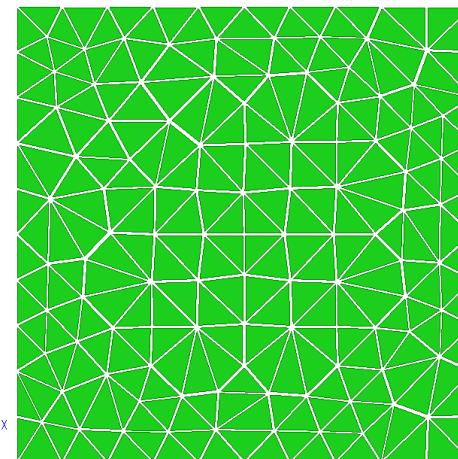
- A regular plate
 - Quad fastest option



Quad



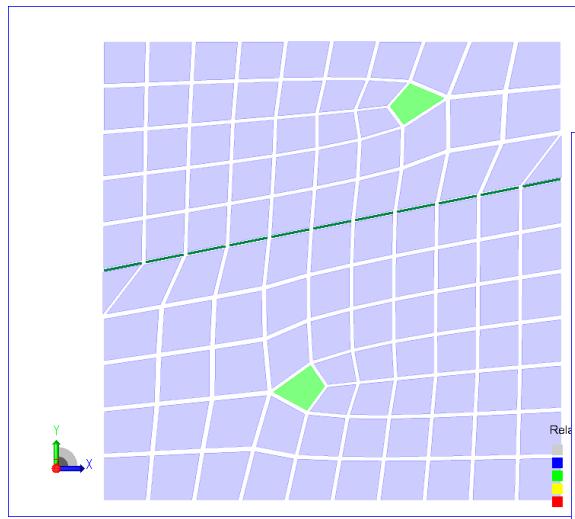
Advancing
front quad



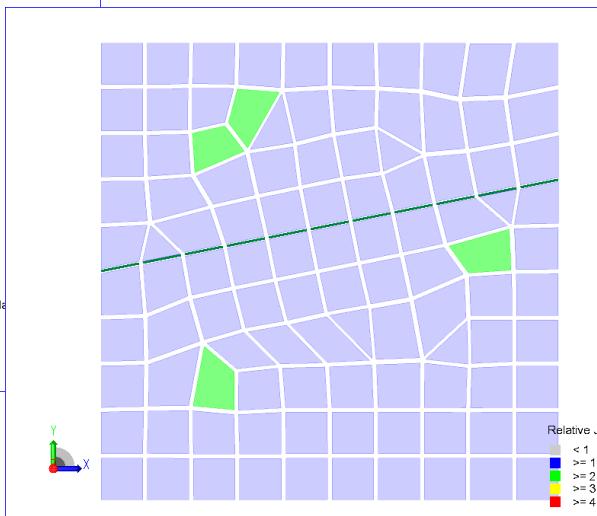
Advancing
front triangle

Quad vs. paver meshing

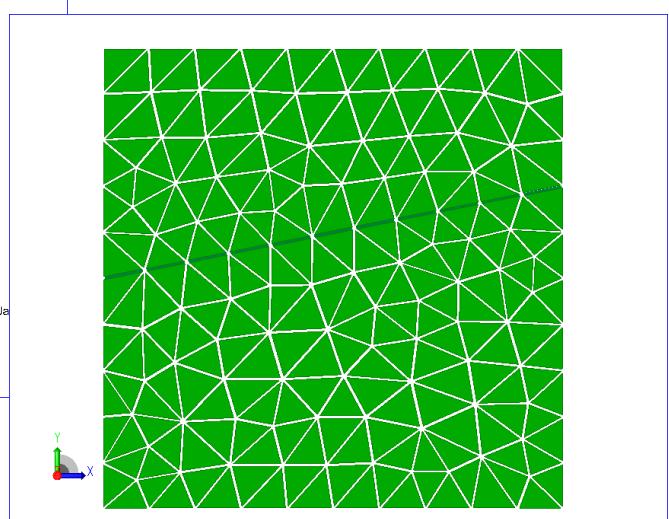
- A regular plate intersected with one beam
 - Quad fastest and seems the best



Quad



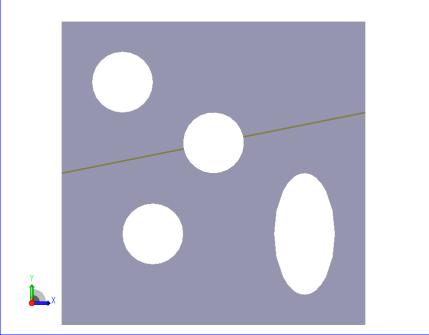
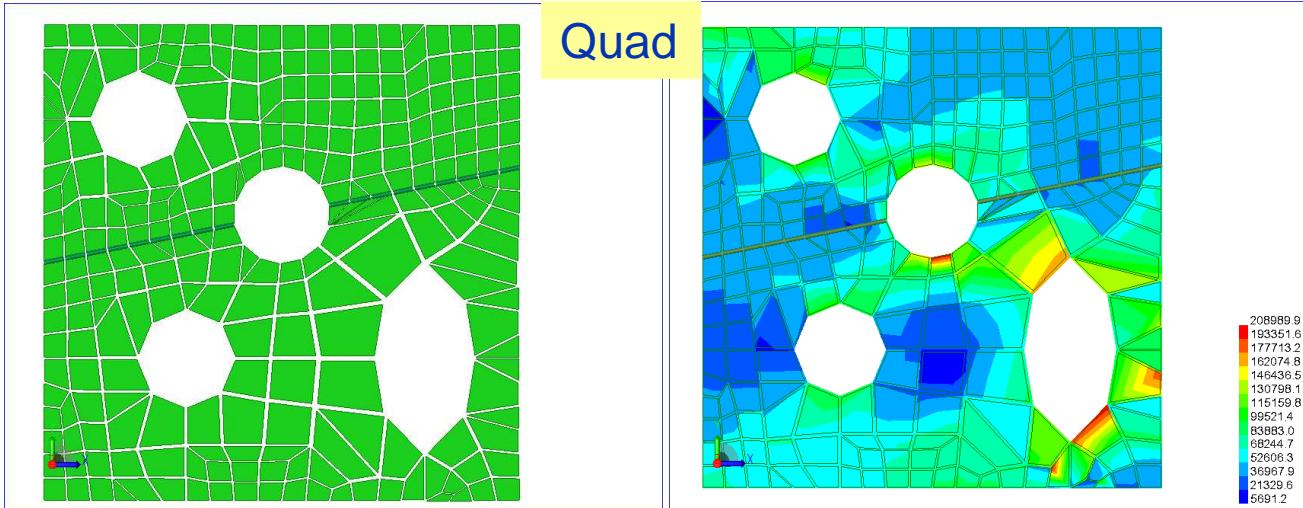
Advancing front quad



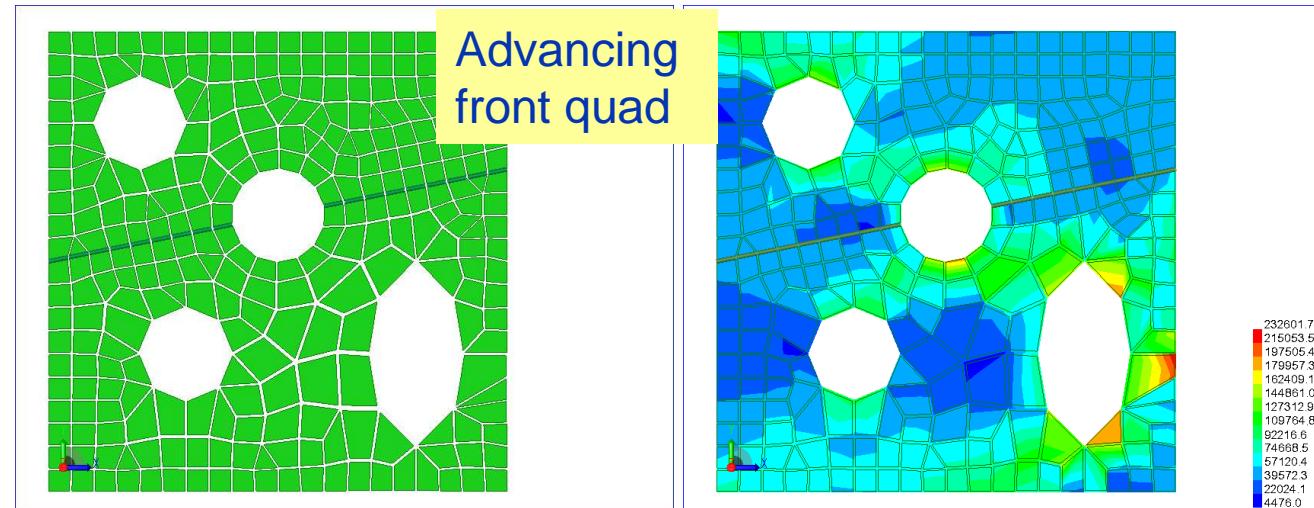
Advancing front triangle

Quad vs. paver meshing

- A regular plate intersected with one beam and four holes



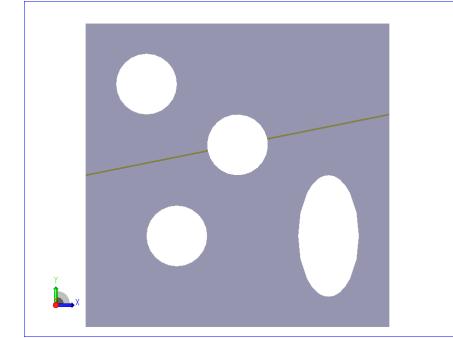
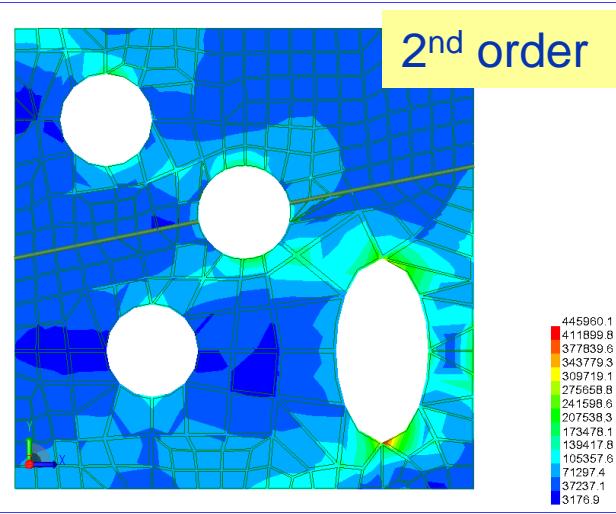
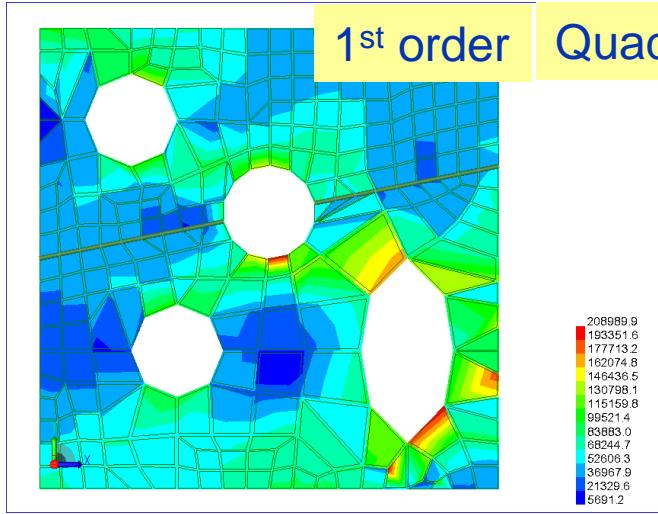
Quad meshing inserts patches to be able to mesh the surface – this deteriorates the mesh in this case



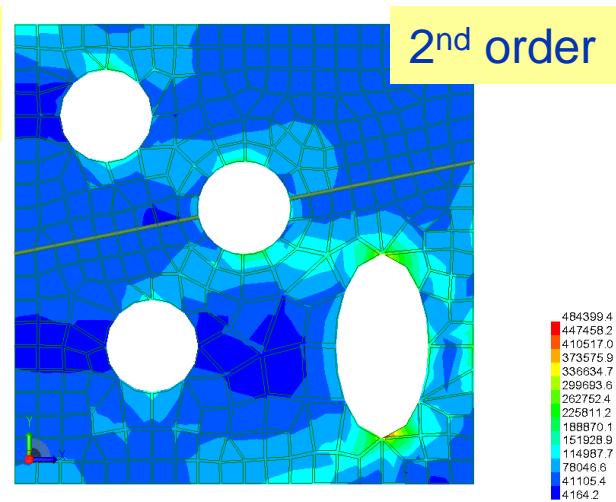
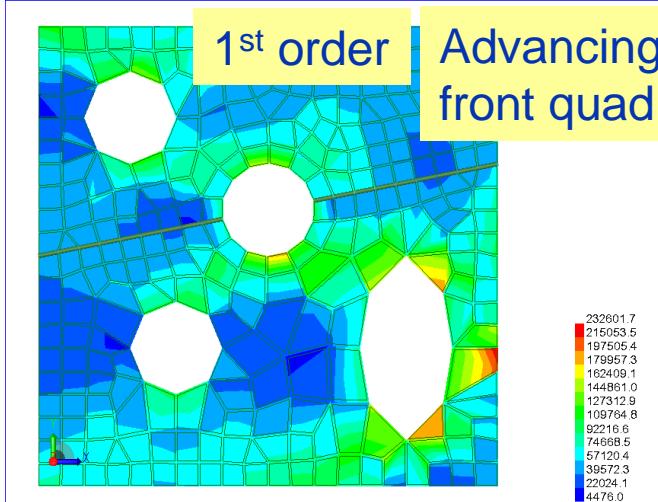
Paver meshing the obvious choice

Quad vs. paver meshing

- The importance of first and second order elements



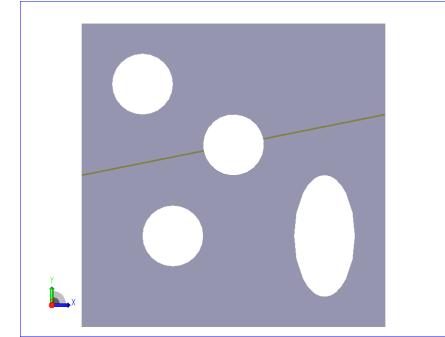
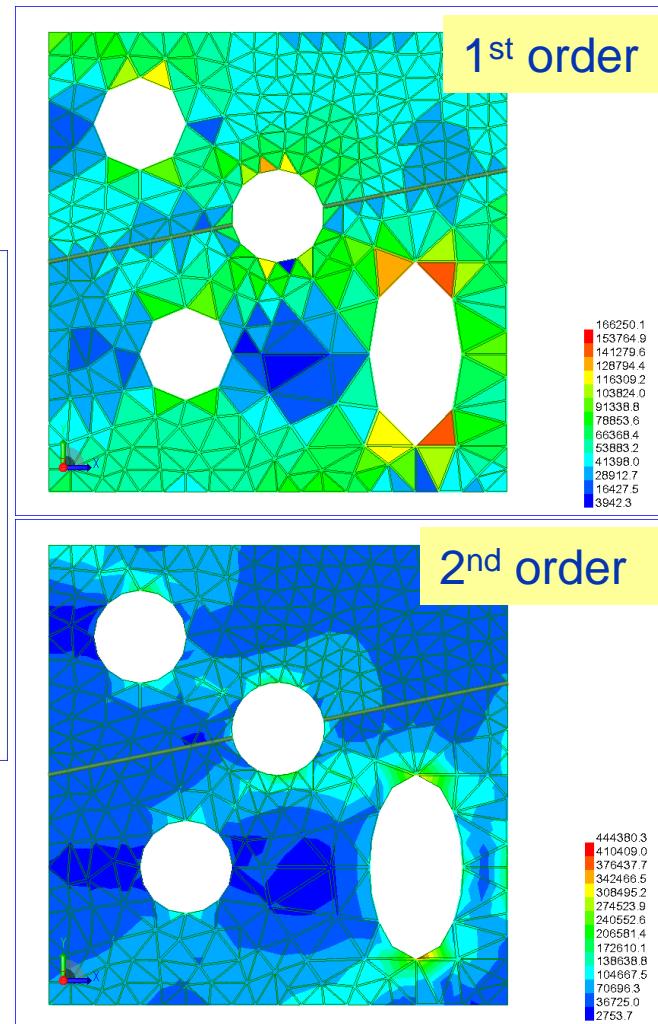
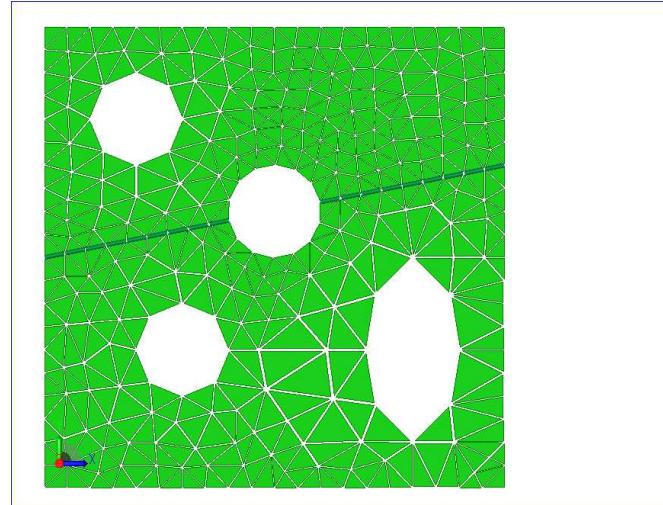
2nd order much better than 1st order for this case



Paver meshing the obvious choice

Quad vs. paver meshing

- A remark on the use of triangular elements

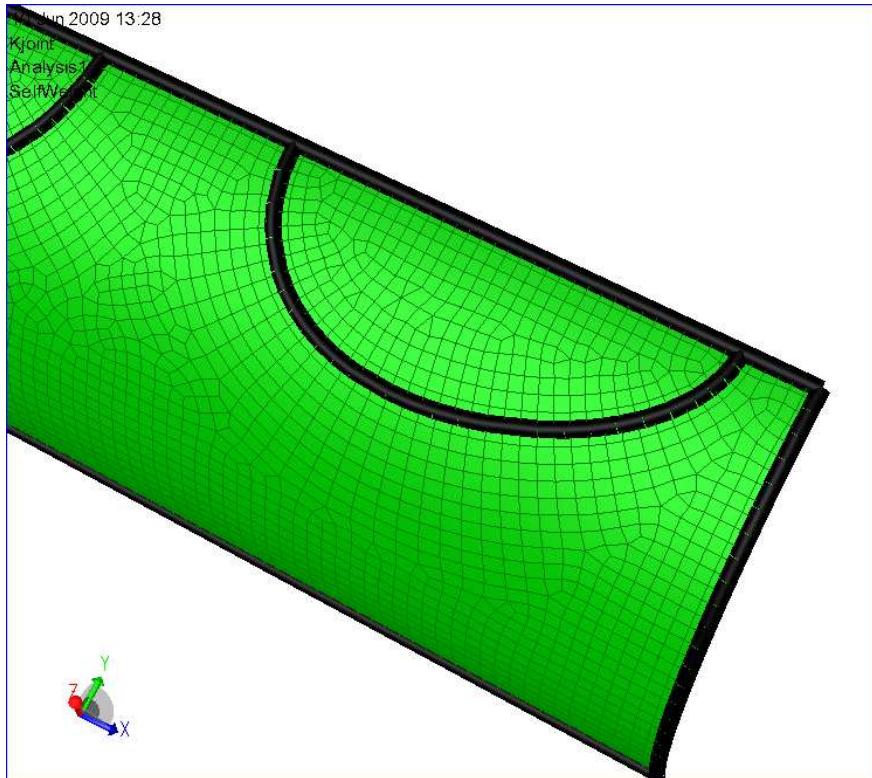


1st order
triangulars are
very stiff elements
and should be
avoided in critical
areas

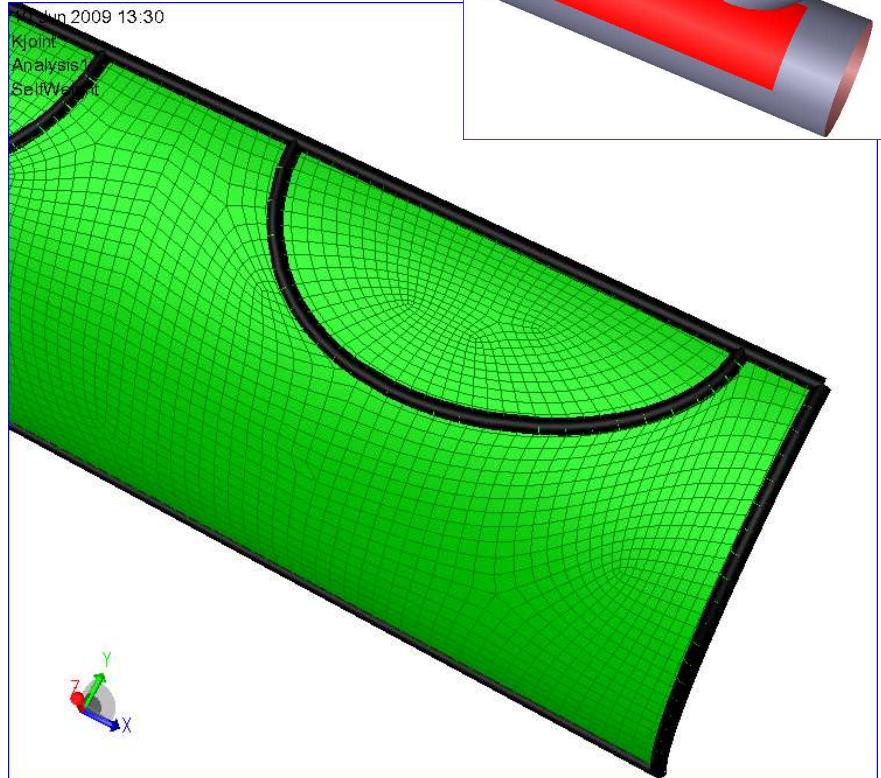
**2nd order much
more reliable**

Quad vs. paver meshing

- A typical example



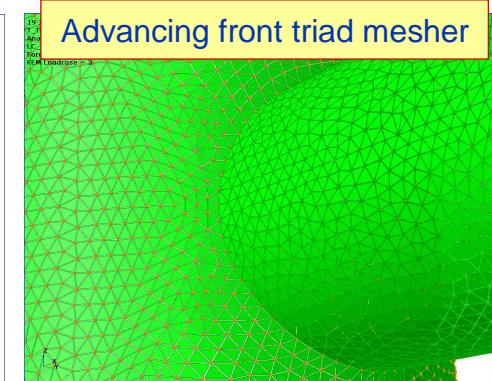
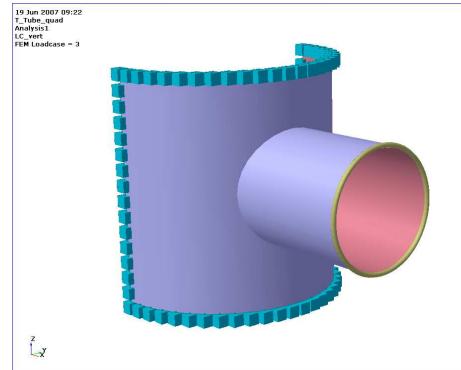
Paver meshing – high quality
mesh along edges



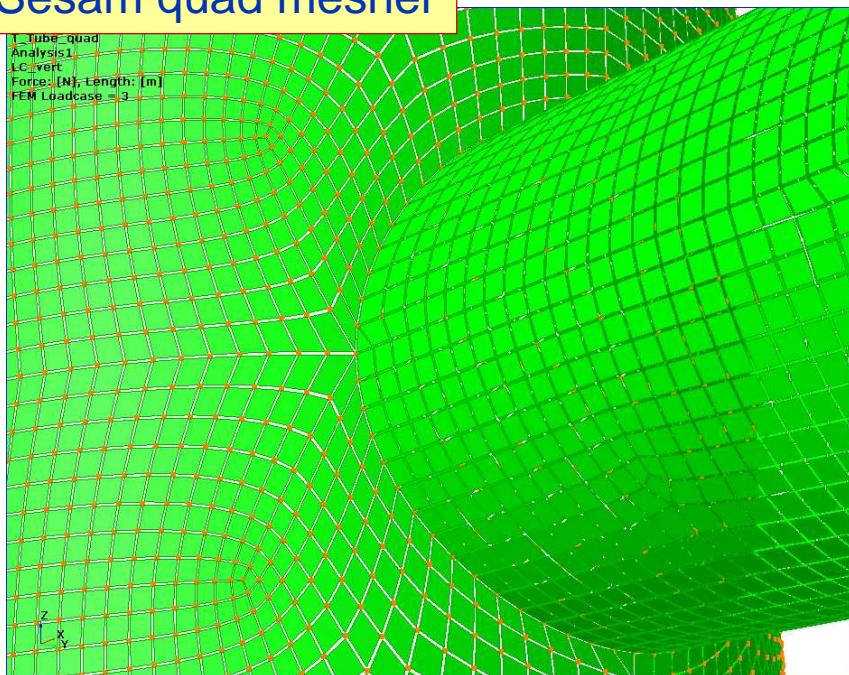
Quad meshing – high quality mesh
in regular patches

Quad vs. paver meshing

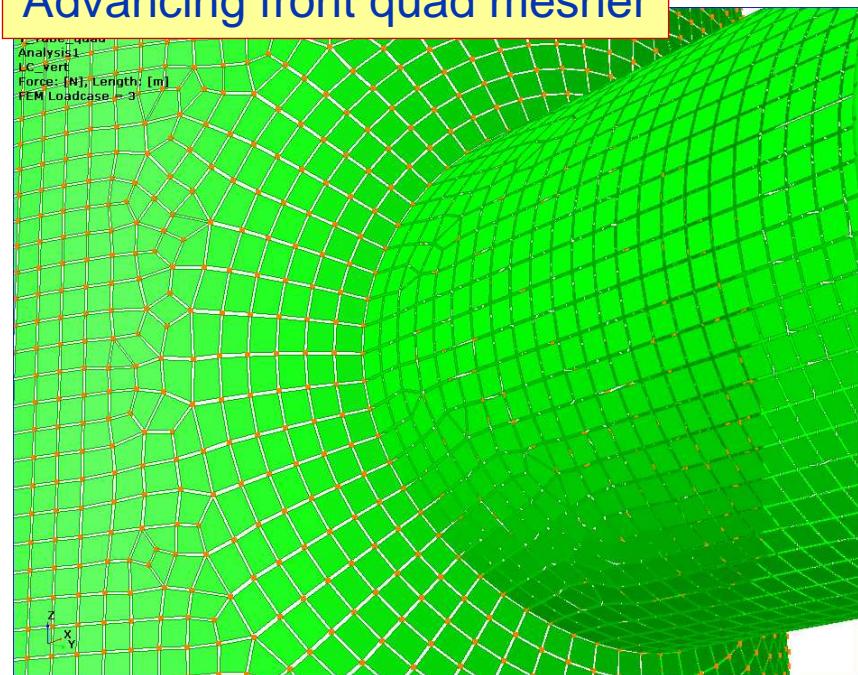
- T-tube to explain differences
 - Global mesh density 0.25 m
 - Forces applied at end of incoming brace



Sesam quad mesher

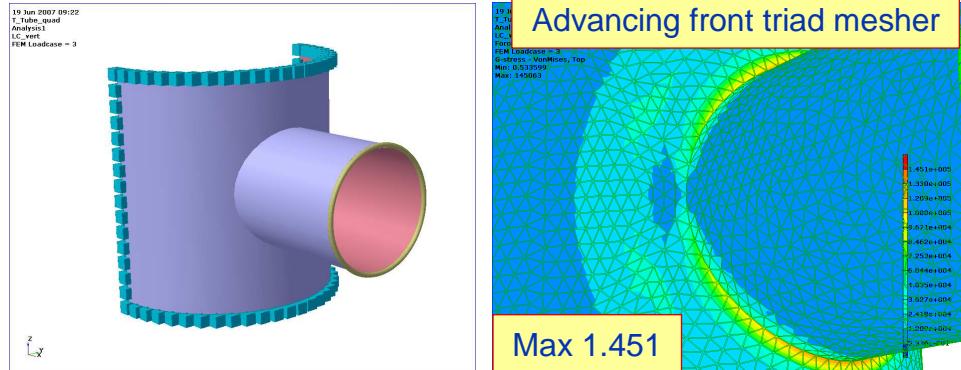


Advancing front quad mesher

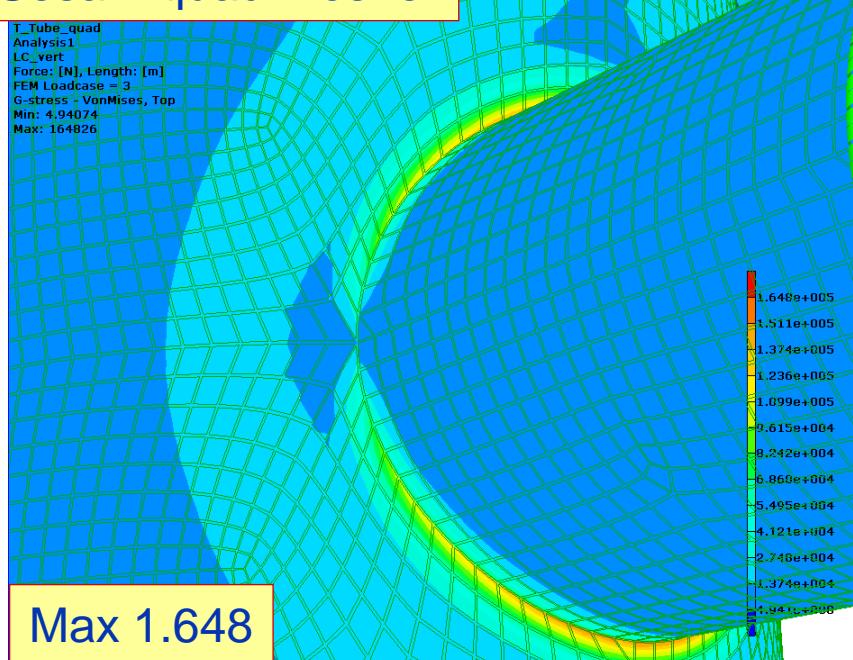


Quad vs. paver meshing

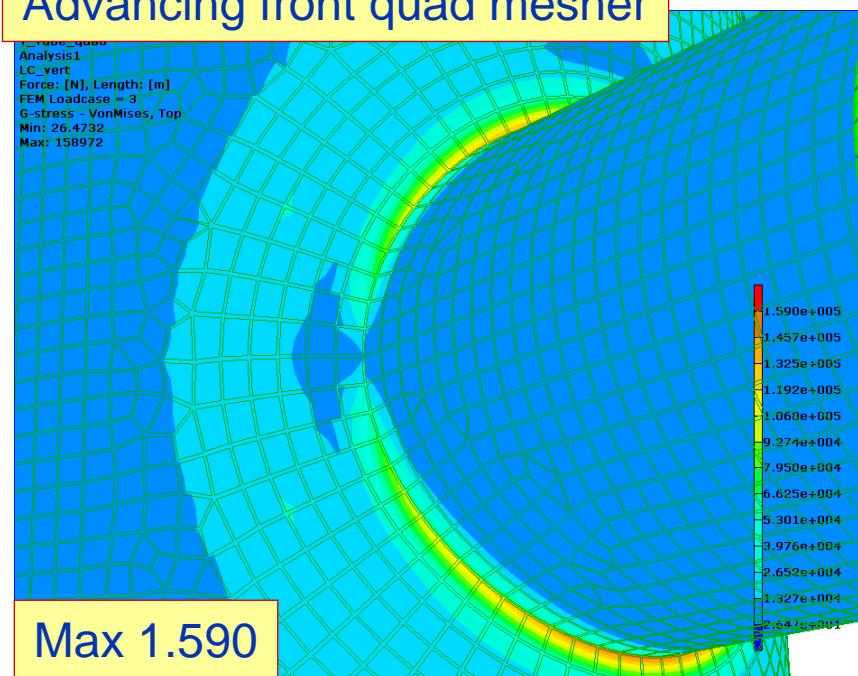
- Paver meshing will make mesh along plate edges first
- Quad mesher will first split plates in patches and then make mesh



Sesam quad mesher

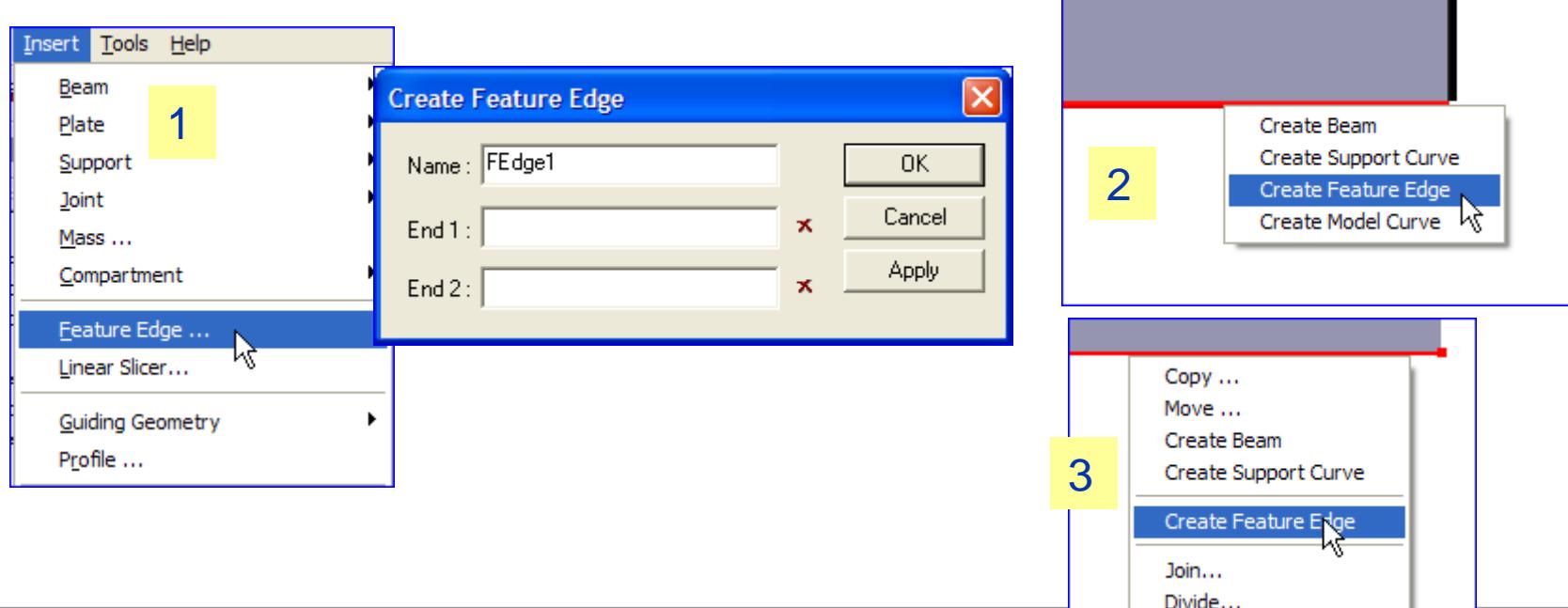


Advancing front quad mesher



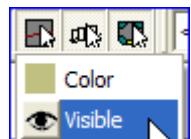
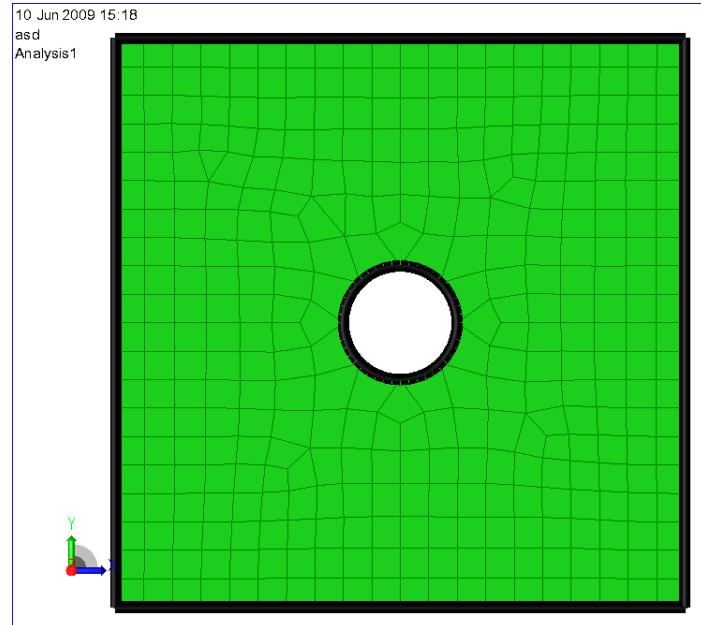
Mesh control lines

- A feature edge will insert mesh line along a surface
 - It must belong to a surface
- Feature edges – or mesh control lines – are activated from
 - 1. *Insert/Feature Edge*
 - 2. Double click surface, select edge, RMB and choose **Create Feature Edge**
 - 3. Select a curve, RMB and choose **Create Feature Edge**



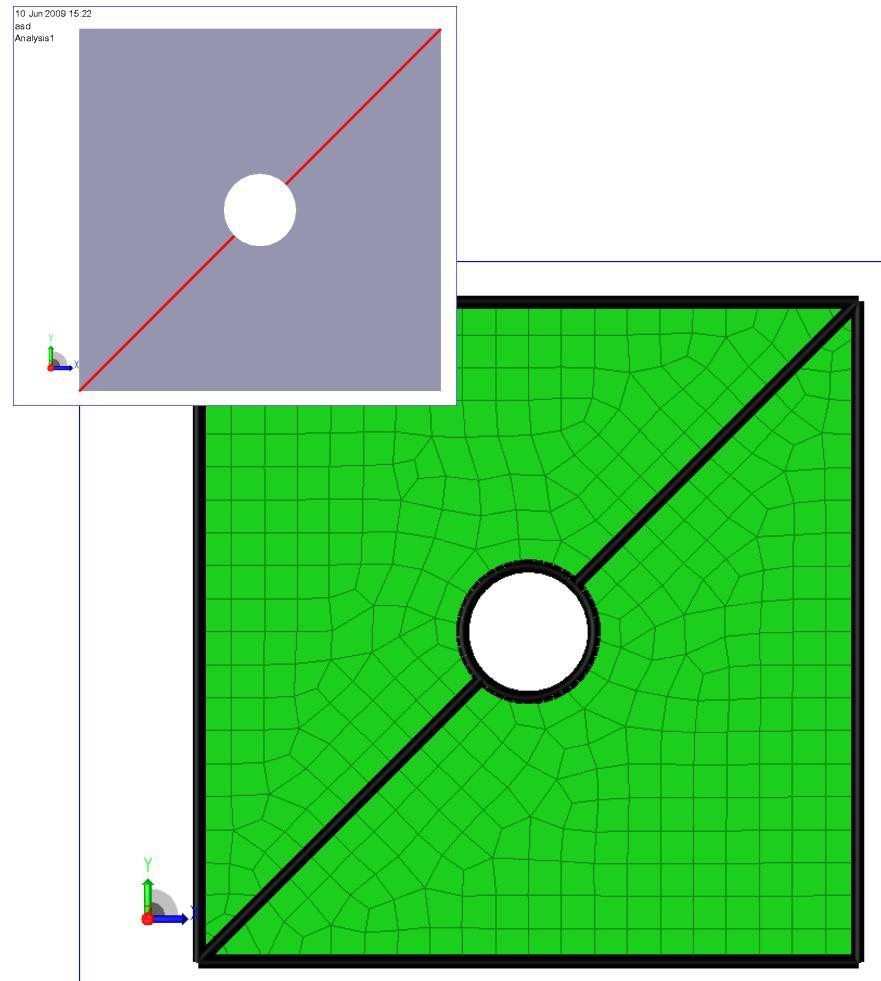
Mesh control lines

- Consider the mesh in a surface with a hole



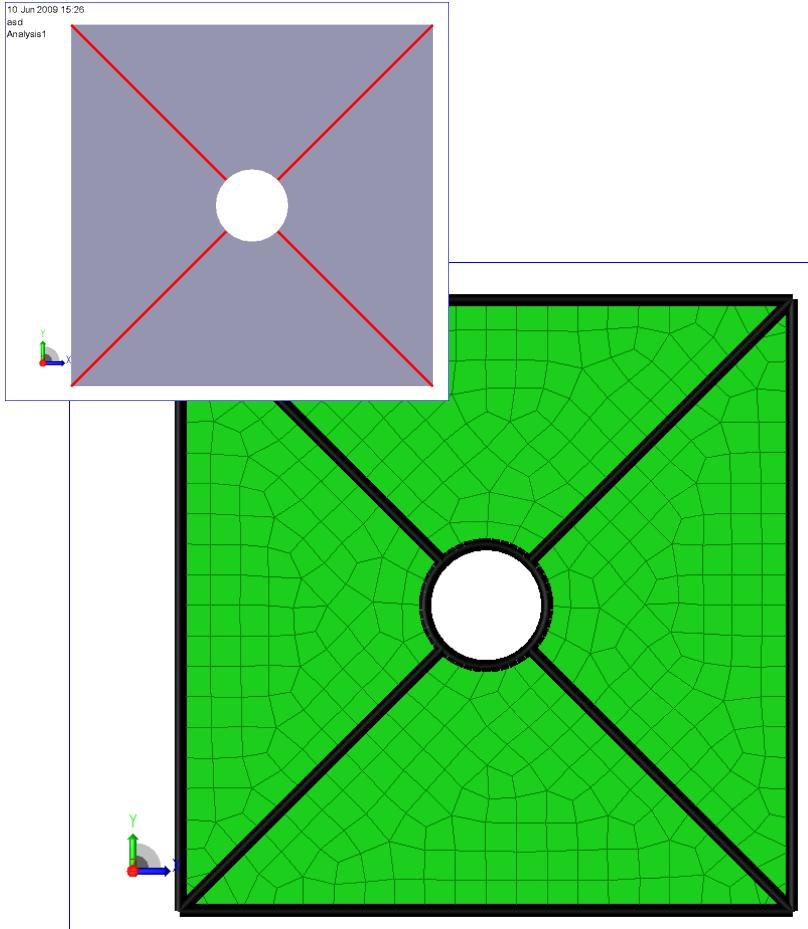
Must be enabled to see feature edges

- Insert a feature edge

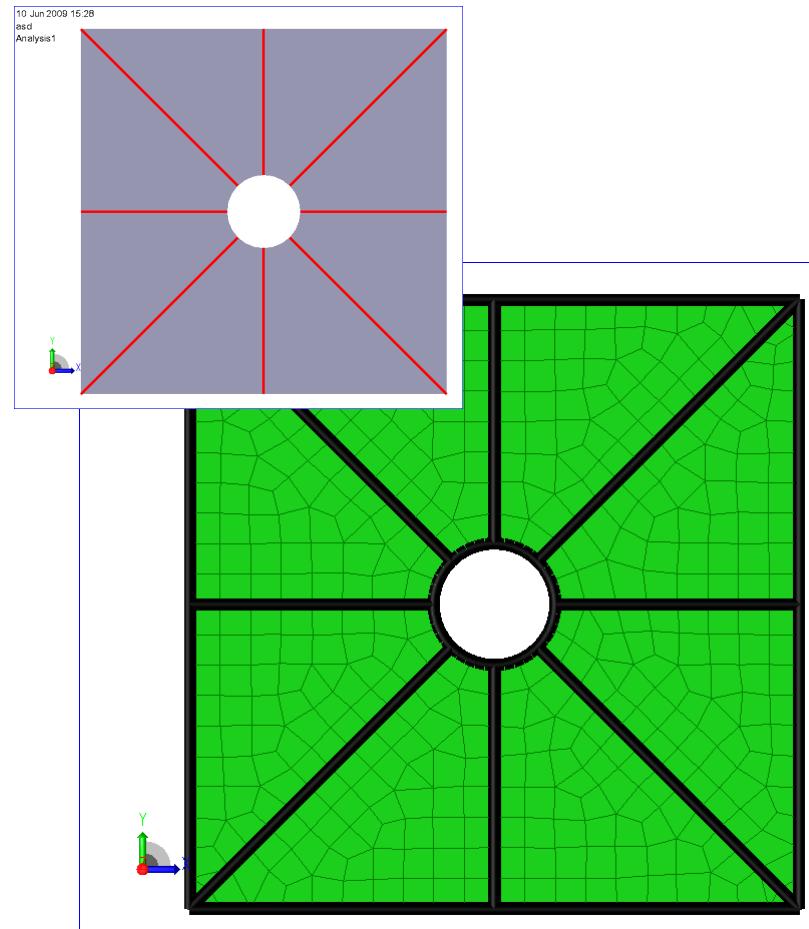


Mesh control lines

- Insert another feature edge



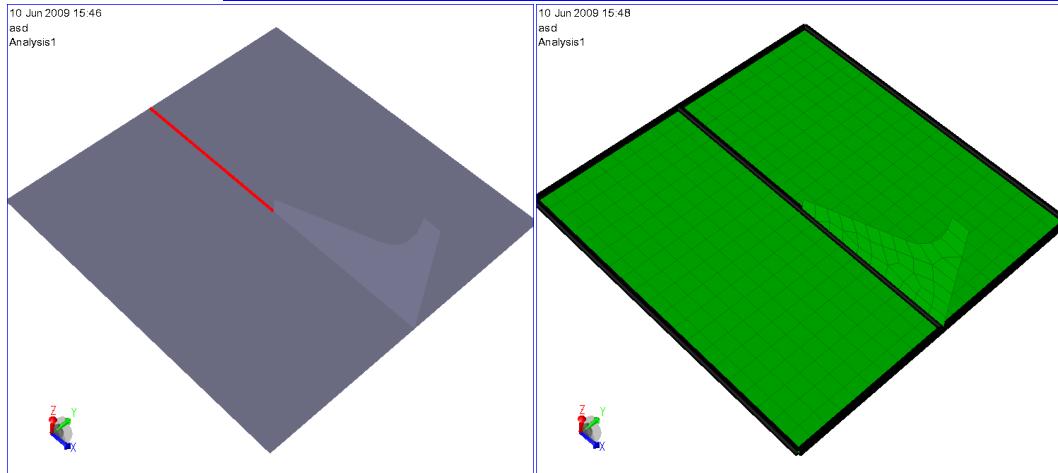
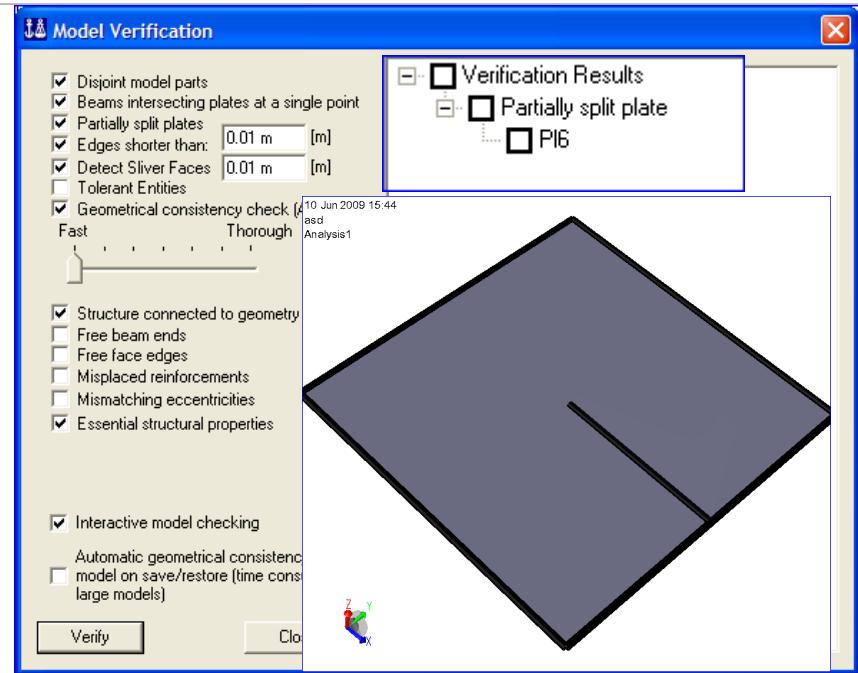
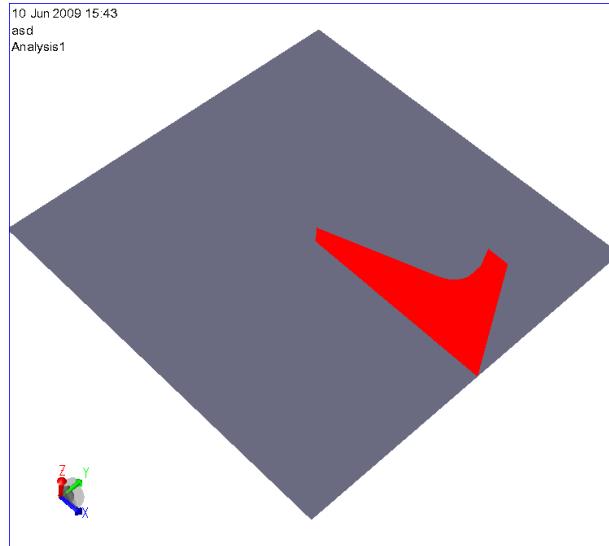
- Insert more feature edges



A feature edge will insert a topological edge

Mesh control lines

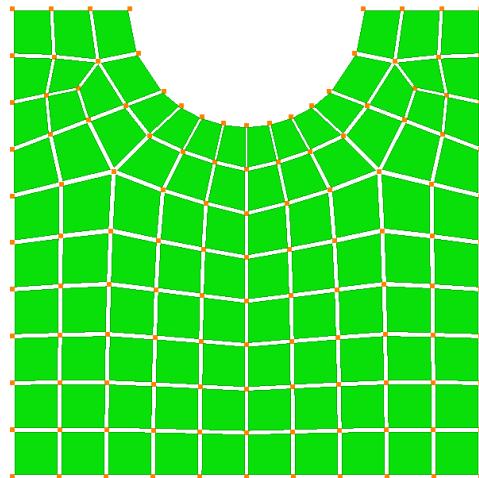
■ Common usage of feature edges



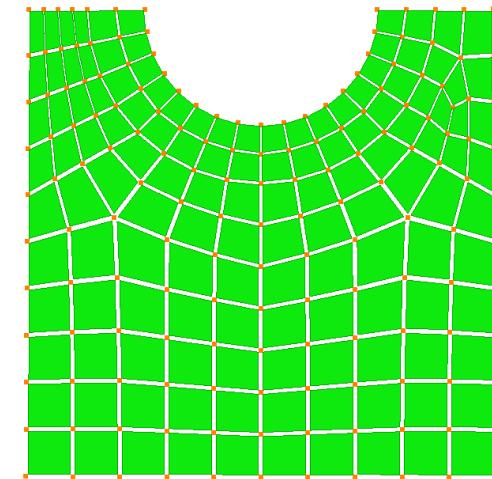
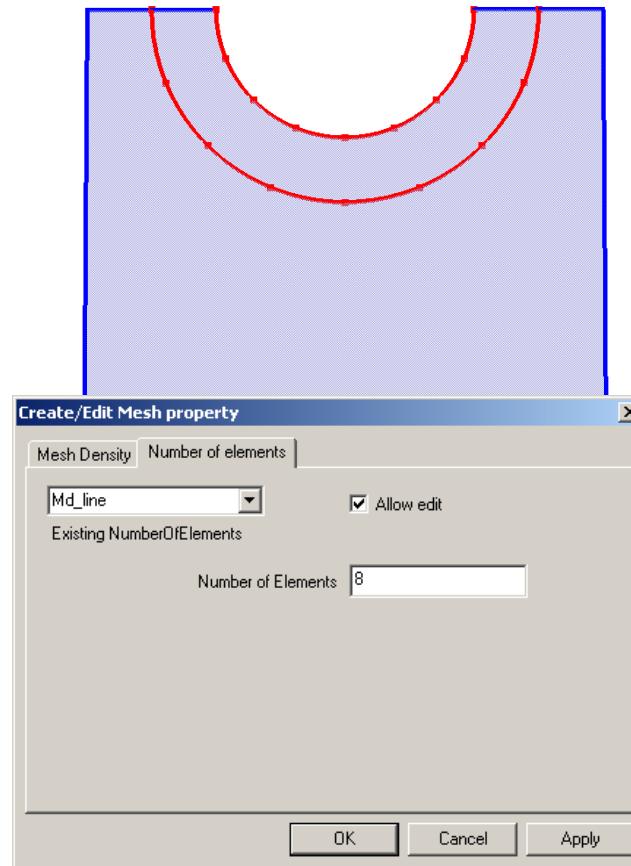
Inserting a feature edge will
insert an edge making a sound
topological model

Mesh control lines

- Apply mesh settings to curved feature edges if you want to control mesh and still use quad meshing



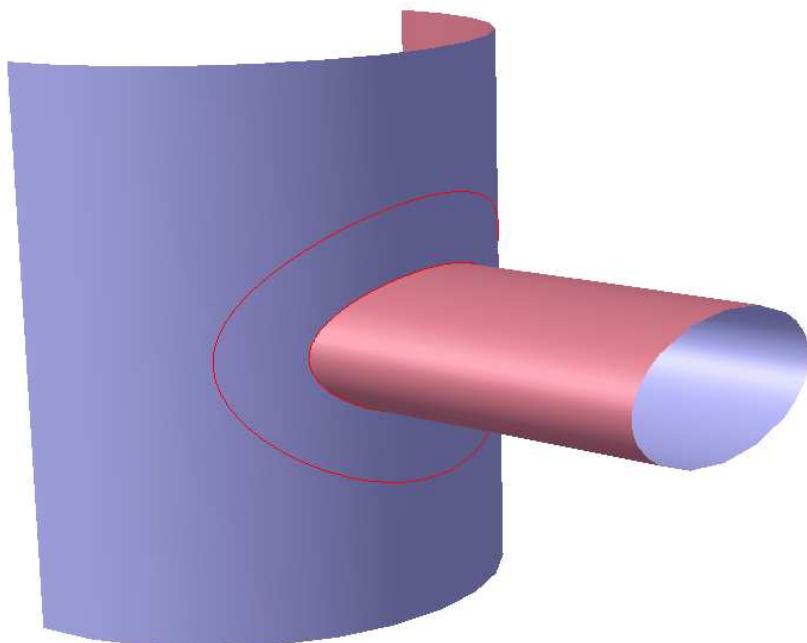
Mesh created by quad meshing



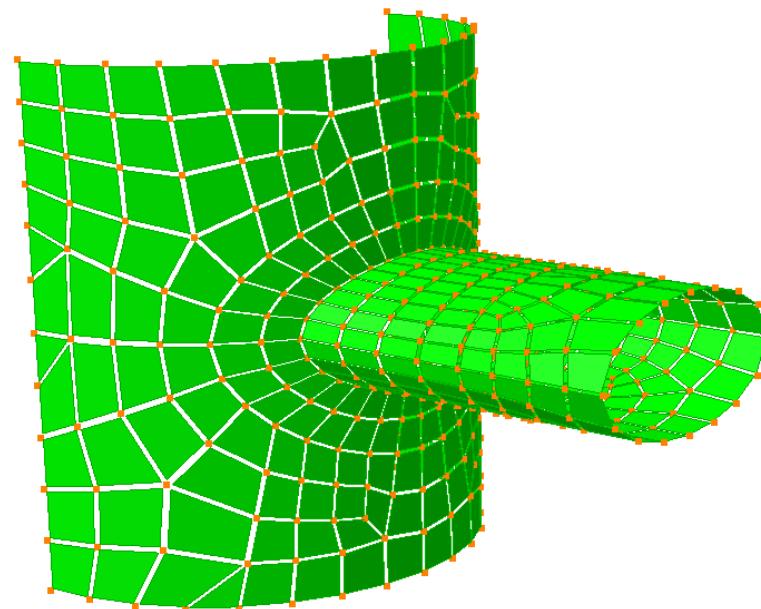
Very good control around hole

Mesh control lines

- Apply mesh control to feature edges based on model curves if you want to control mesh and still use quad meshing



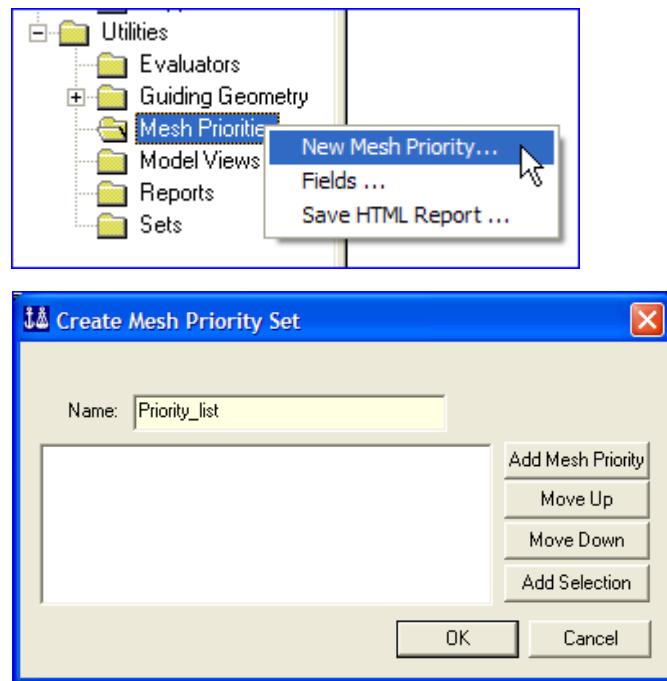
In this case a dummy brace is used to create the model curve on the chord



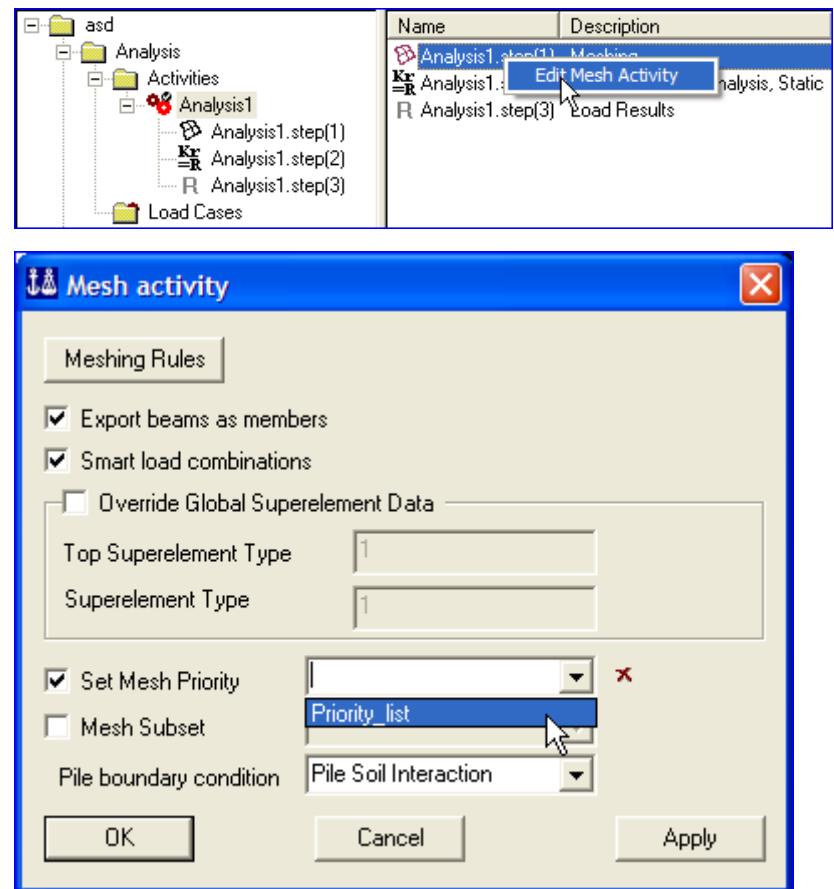
8 elements along each quadrant
i.e. 32 elements around the circumference

Prioritized meshing

- The sequence of meshing may be defined and is activated from a mix of mesh priorities and using them in the mesh generation
- The mesh priorities are defined from



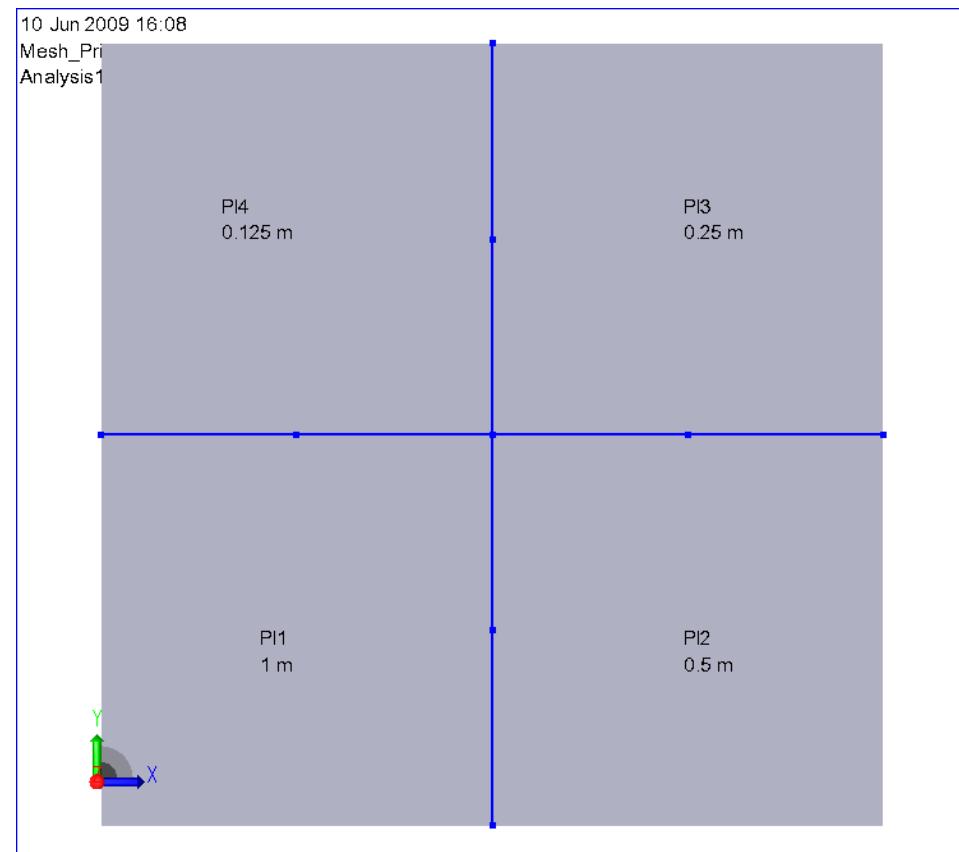
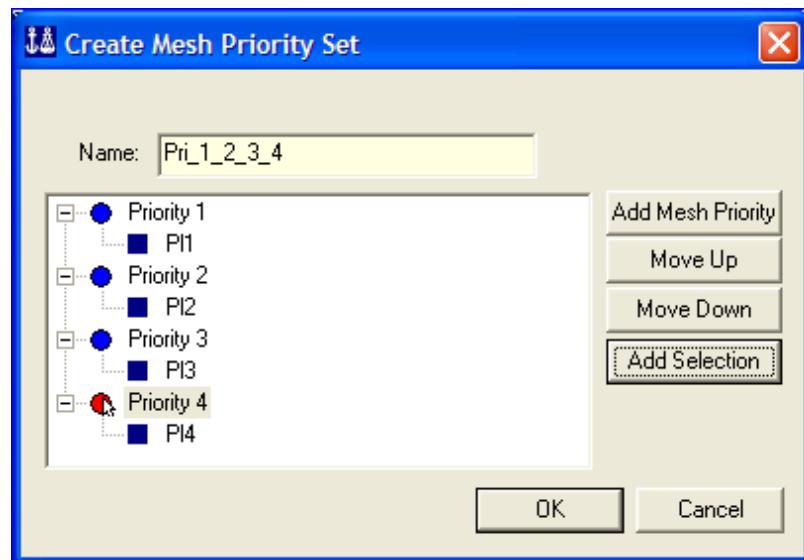
- Mesh priorities are used when activated from the mesh settings in the analysis activity



Prioritized meshing

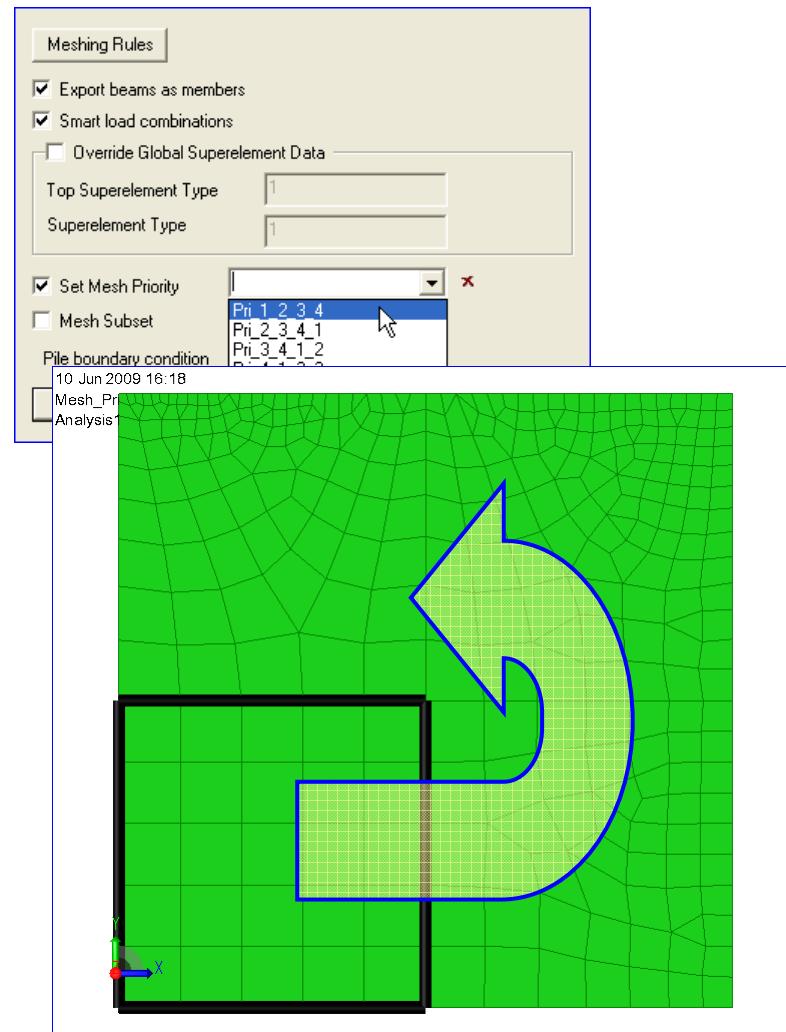
- Explaining mesh priorities by an example

- Pri_1_2_3_4
- Pri_2_3_4_1
- Pri_3_4_1_2
- Pri_4_1_2_3
- Pri_4_AND_2

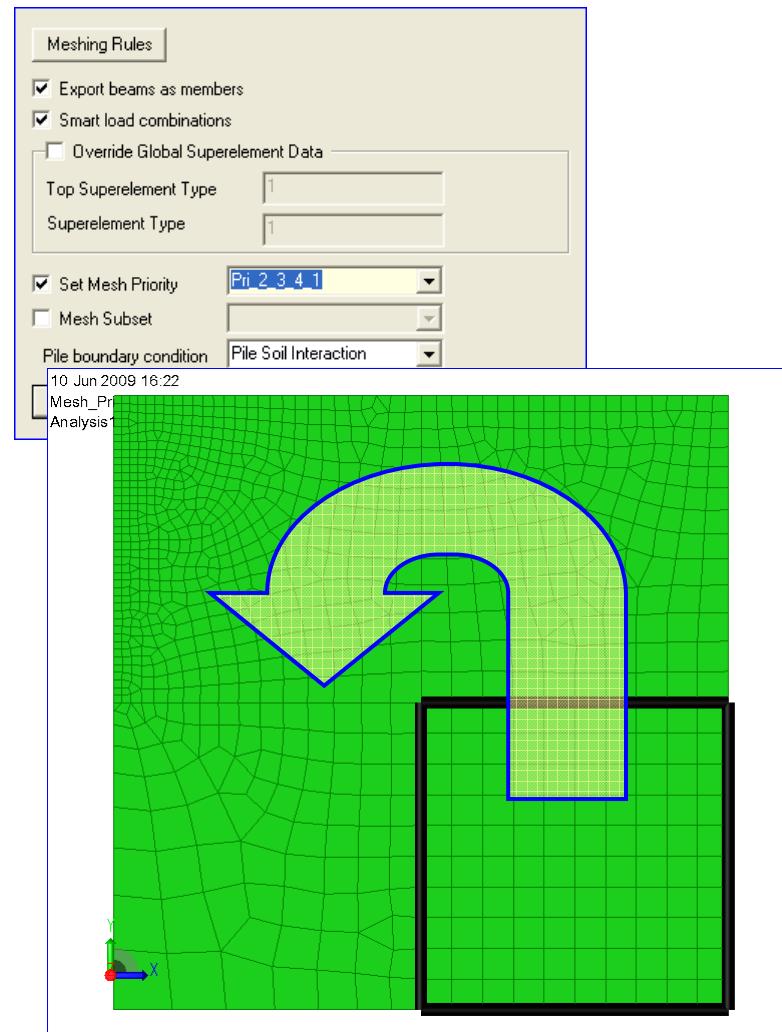


Prioritized meshing

■ Pri_1_2_3_4

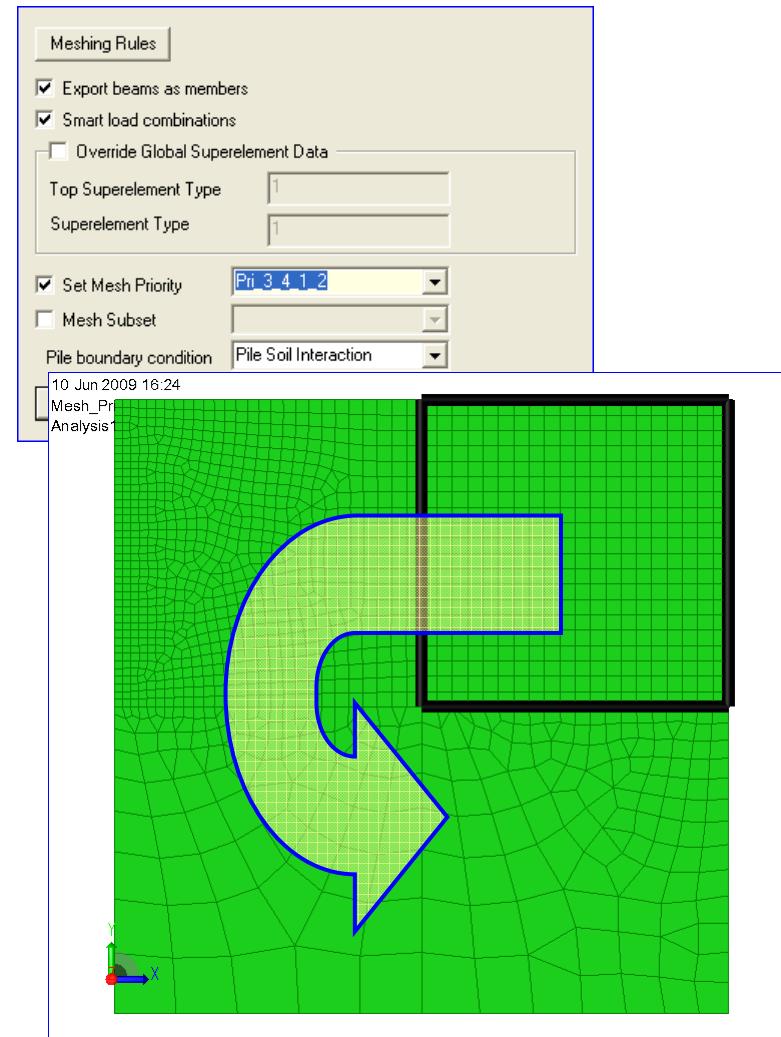


■ Pri_2_3_4_1

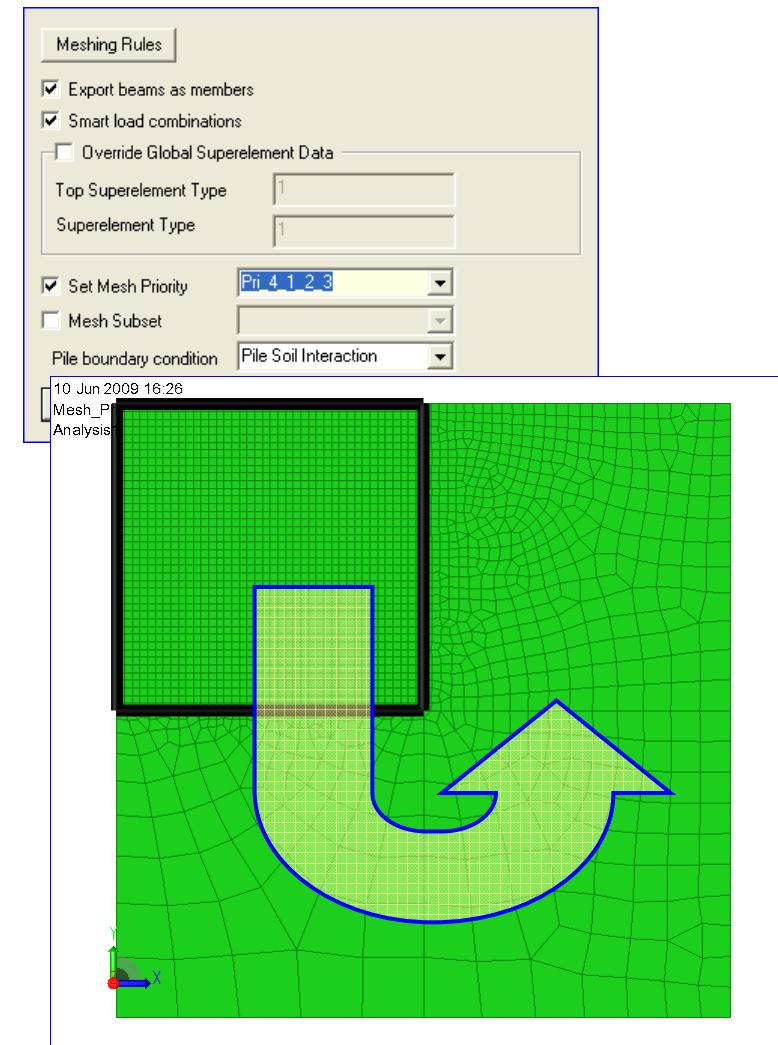


Prioritized meshing

■ Pri_3_4_1_2

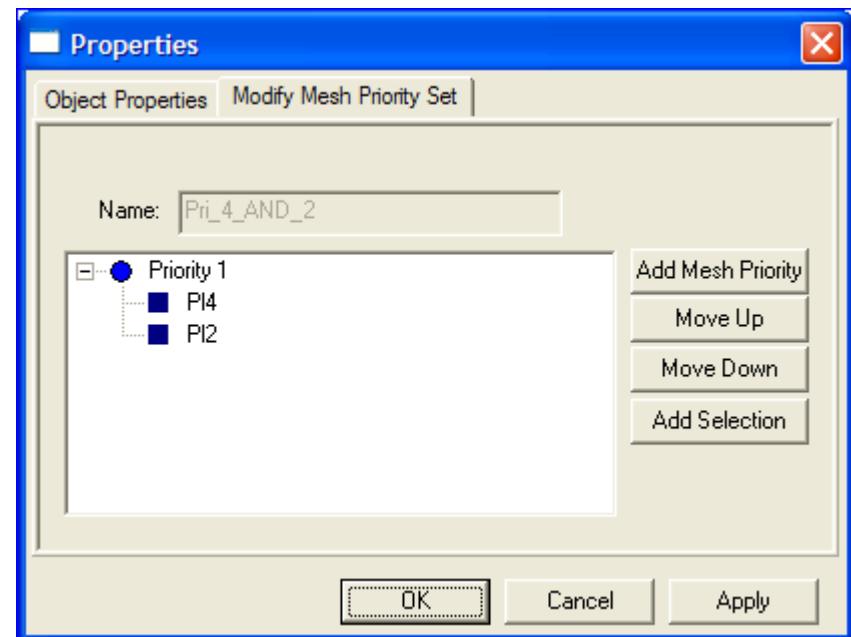
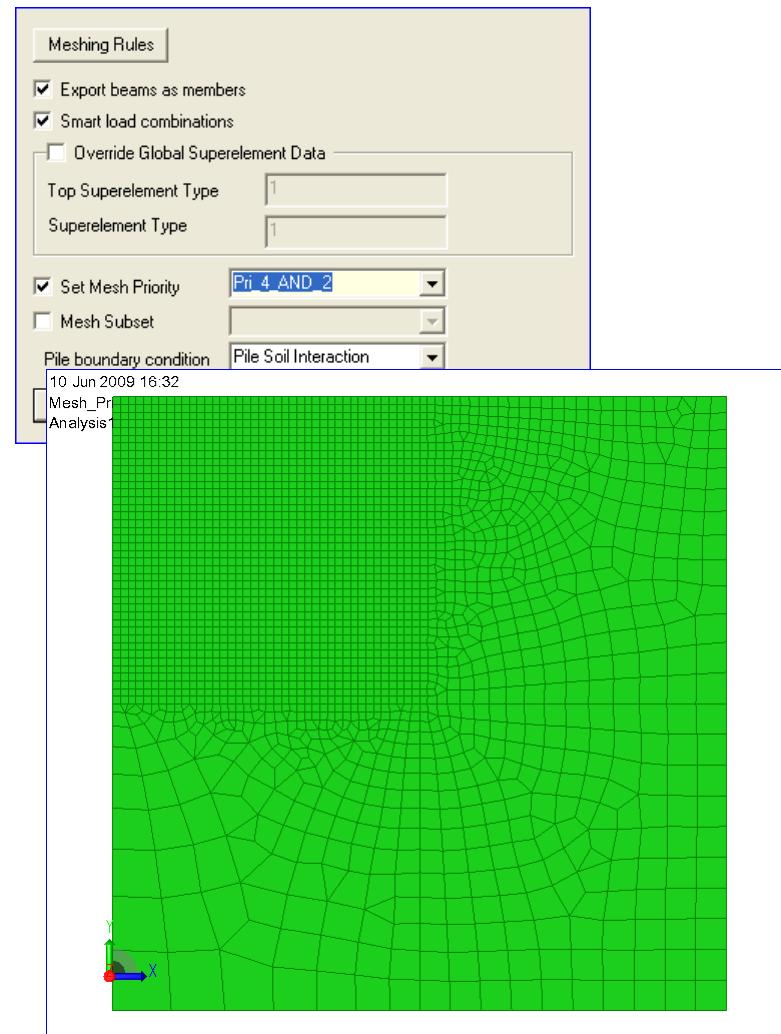


■ Pri_2_3_4_1



Prioritized meshing

■ Pri_4_AND_2

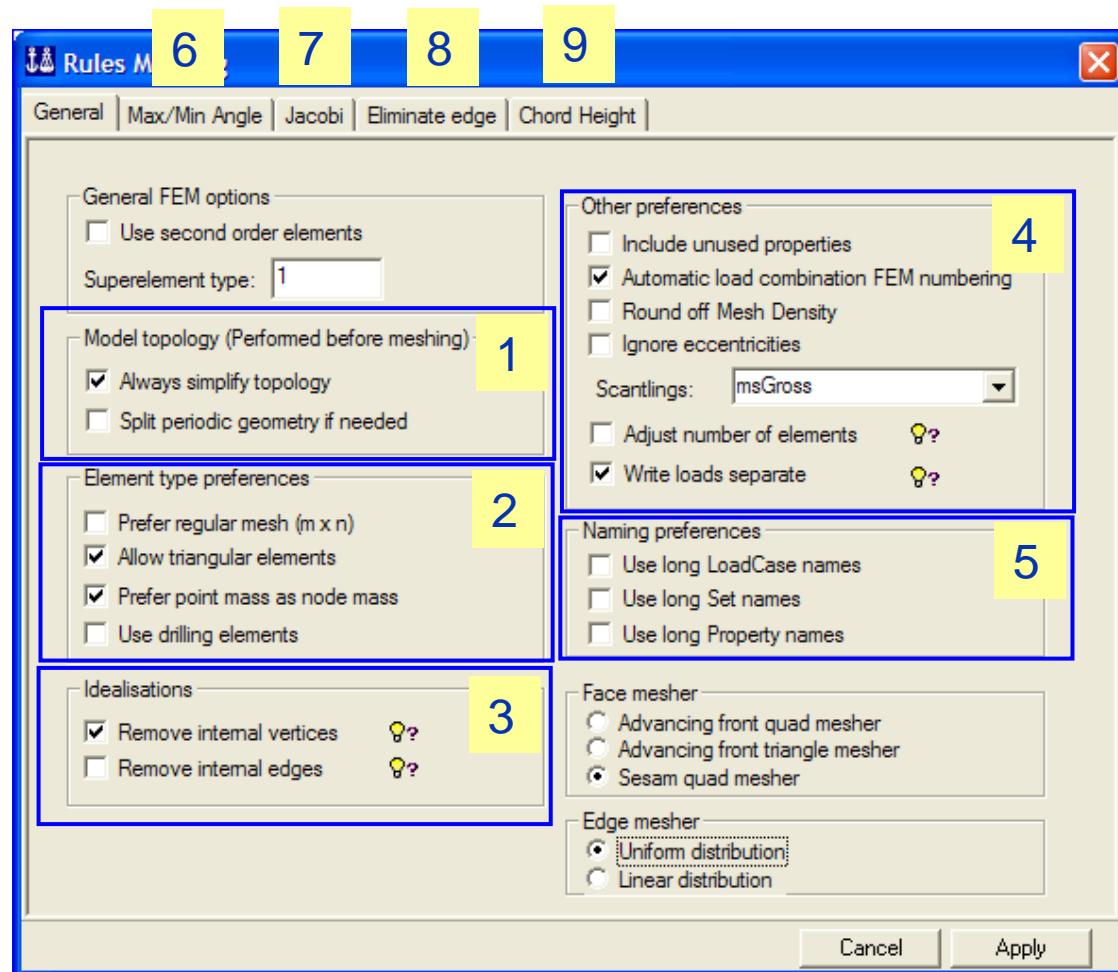


Any structure not part of a prioritized list will be meshed after those on the list

Advanced mesh settings

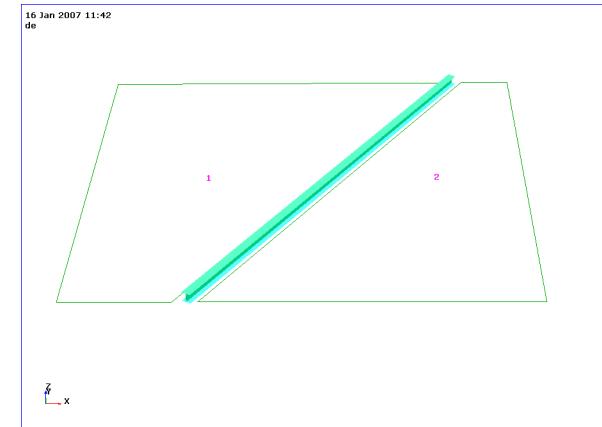
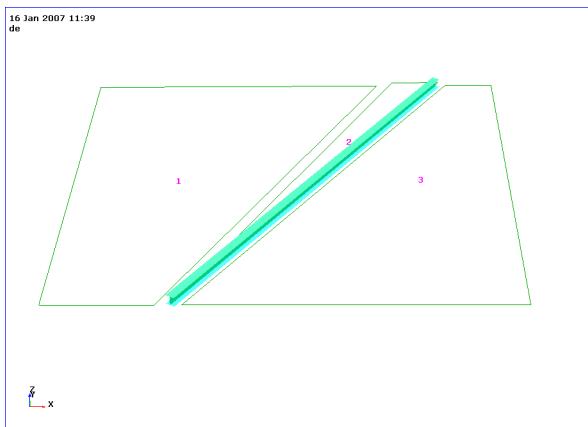
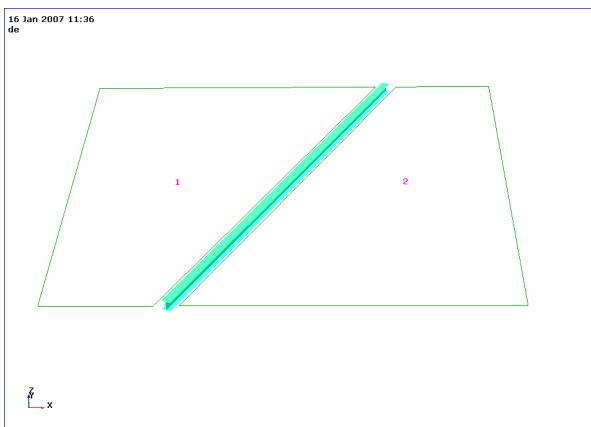
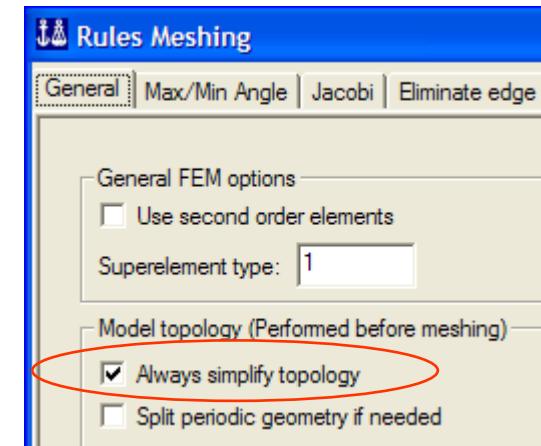
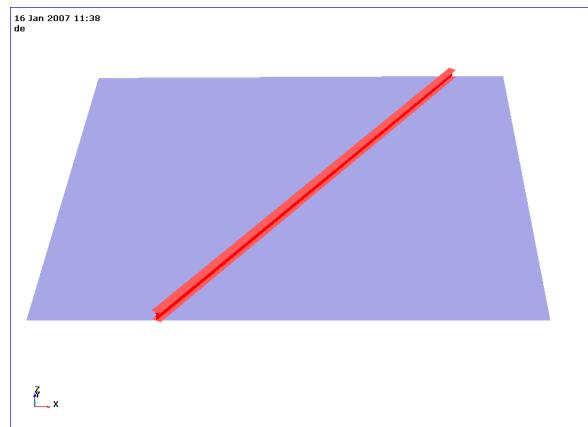
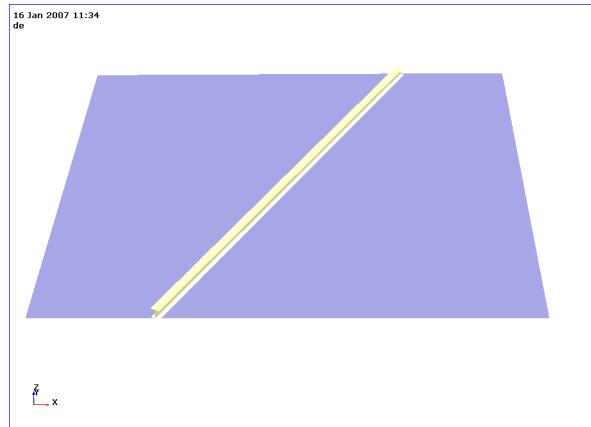
■ A walkthrough of the mesh control options not discussed yet...

- In addition to 1-9:
- 10: Mesh locking
- 11: Make your own program defaults
- 12: Useful tools



Simplify topology

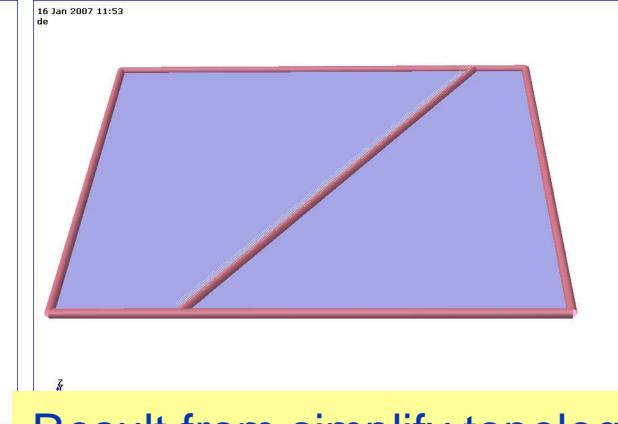
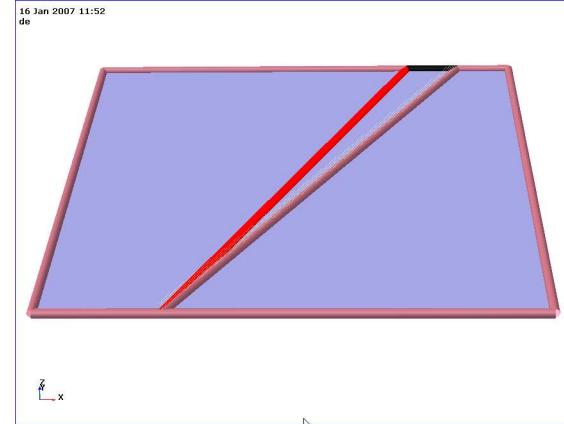
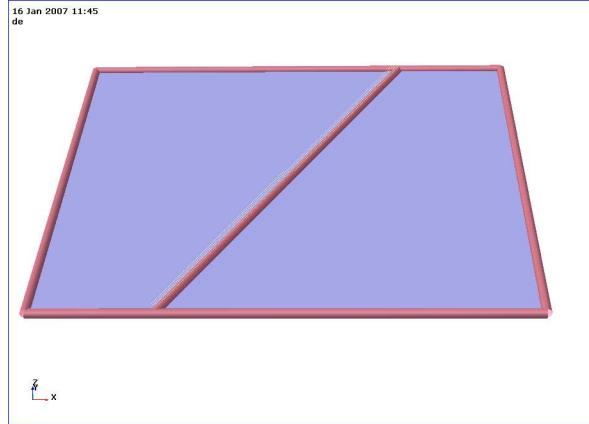
■ Simplify topology



Simplify topology

- Simplify topology – what goes on behind the scene
 - Double-click the plate to get access to topology model
 - You may enter data to the topology lines
- There will always be a mesh line along a topology line
 - Defined by beams, feature edges, connection to other plates, outer edges of plates or internal vertices used to describe e.g. holes

Create Beam
Create Support Curve
Create Feature Edge
Create Model Curve

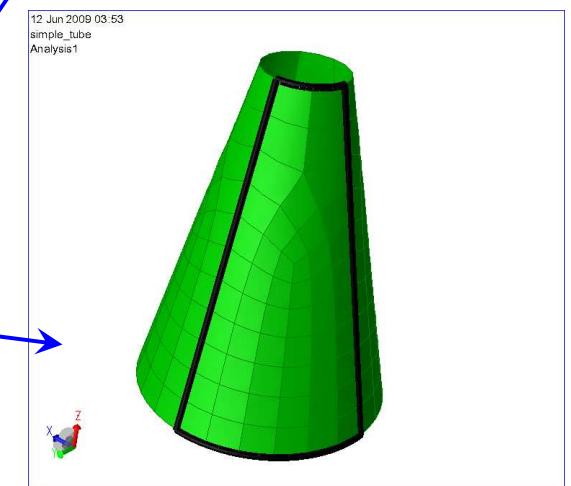
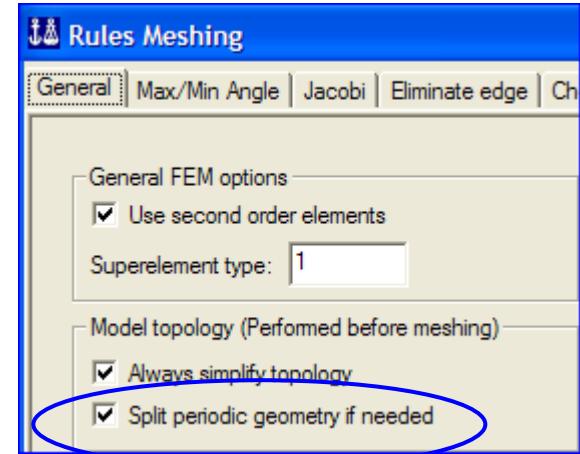
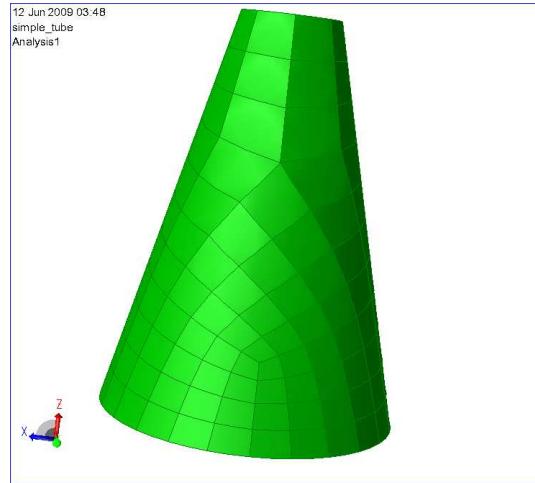
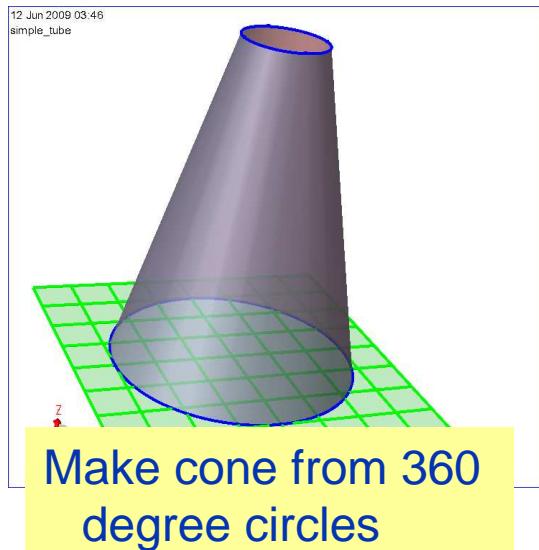


Result from simplify topology

- Simplify topology can also be done manually:
Tools > Structure > Geometry > Simplify Topology

Split periodic geometry if needed

When to use the feature



If above meshing fails use split periodic geometry or divide the cone into e.g. 90 degrees parts

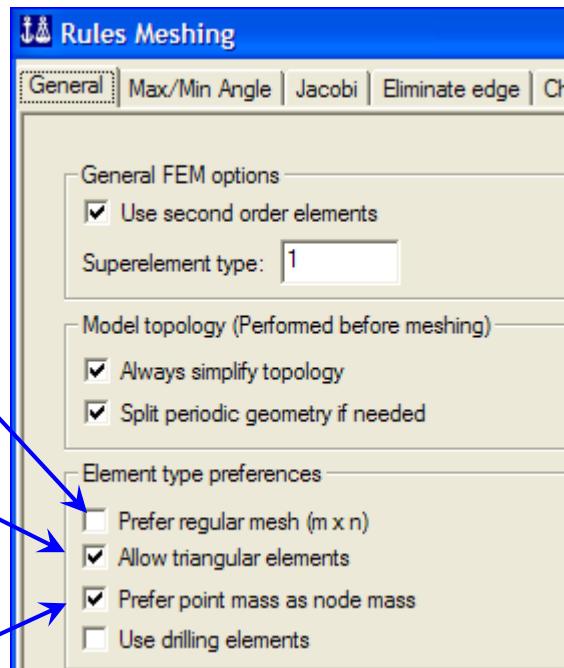
Element type preferences

■ Special options

Use with care, may end up with many rectangular elements

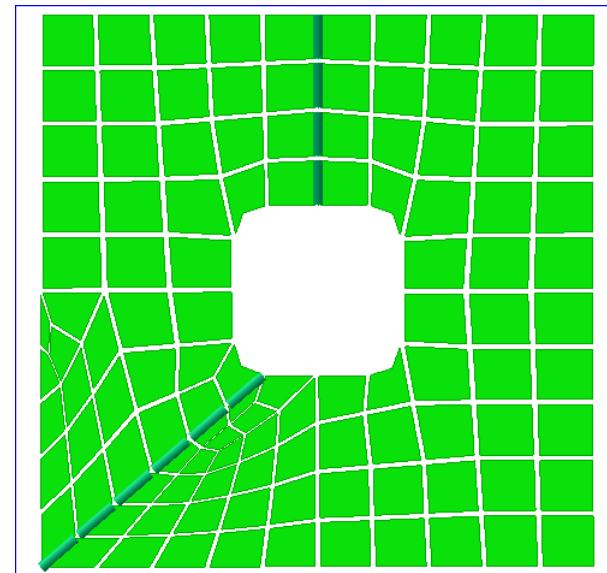
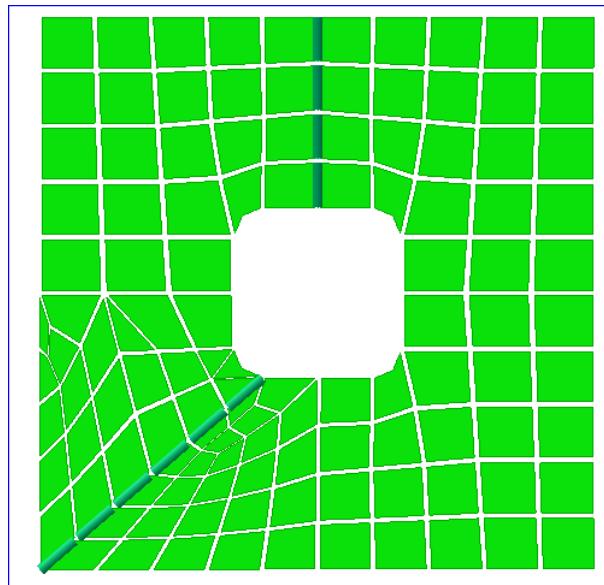
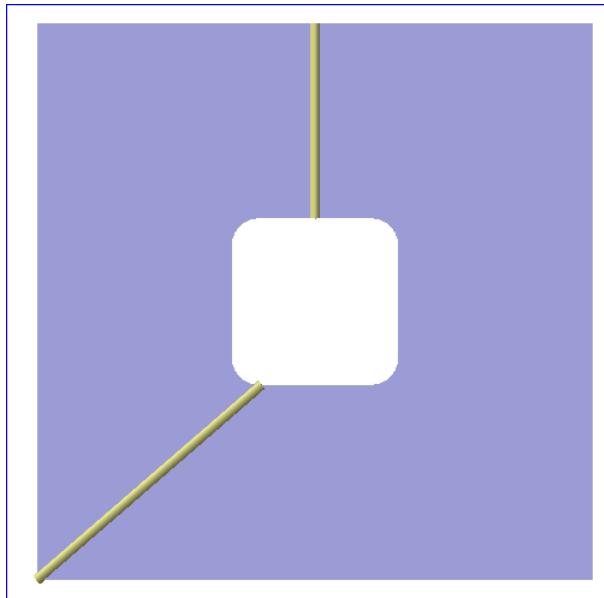
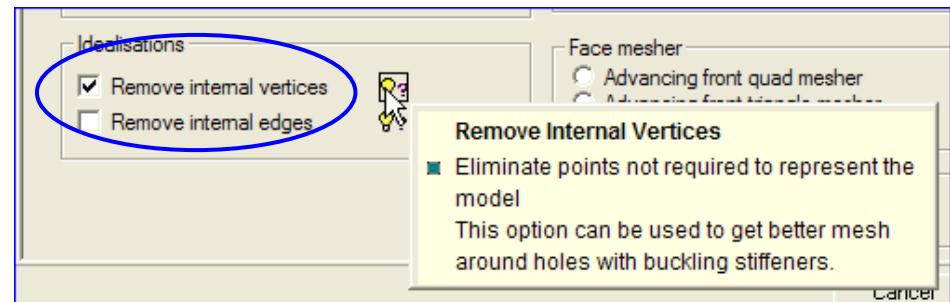
Disable triangular elements, i.e. use as few triangles as possible to make the mesh

Represent point masses as nodal masses? I.e. not inserting a mass element with zero length



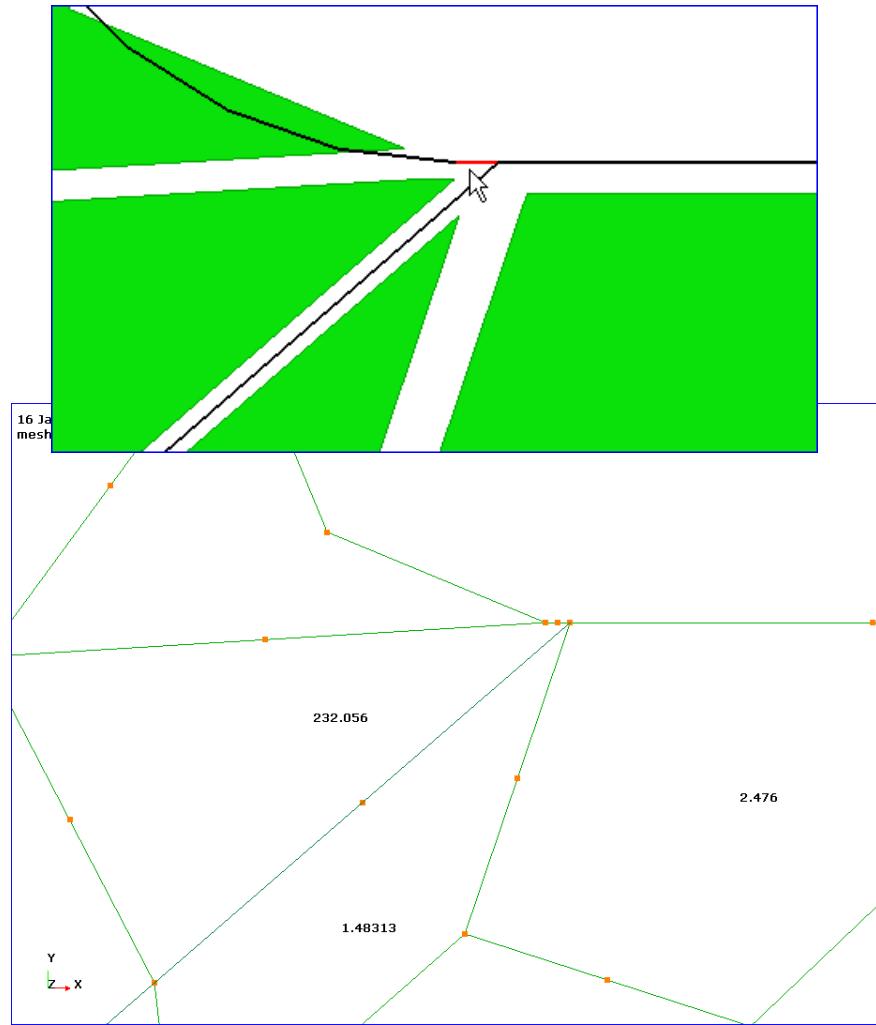
Remove internal vertices

- Will in most cases make a better mesh around holes with buckling stiffeners
 - The default is: Enabled



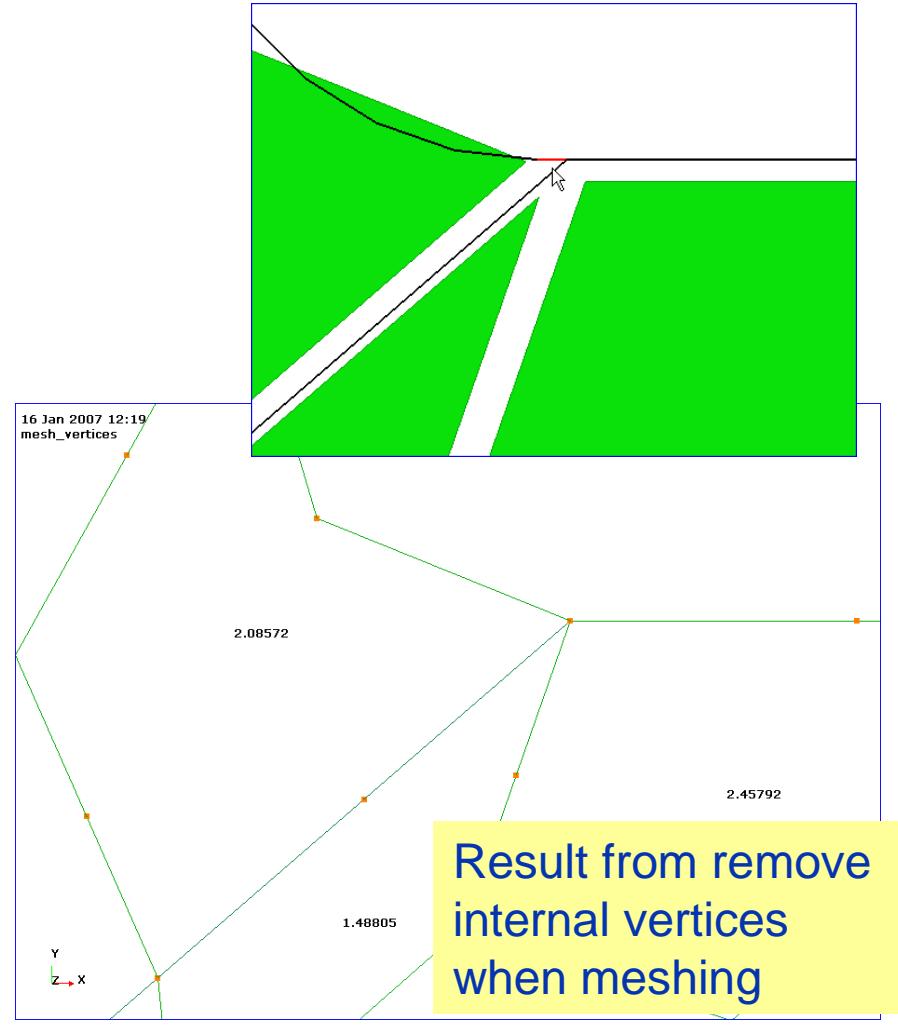
Remove internal vertices

■ Behind the scene



Mesh guidance GenIE v5.1

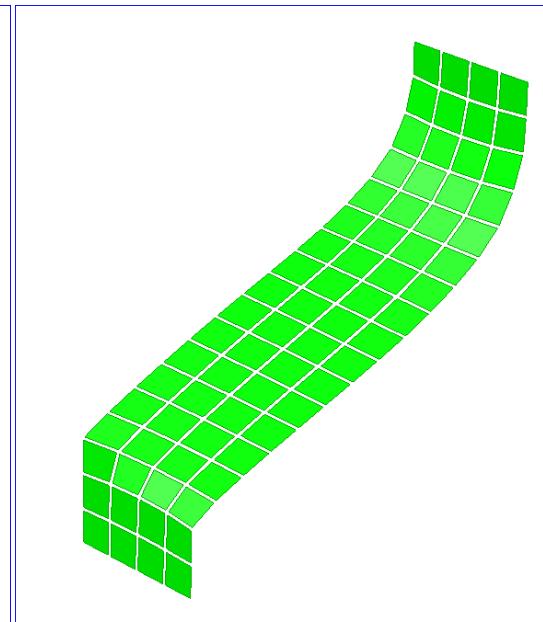
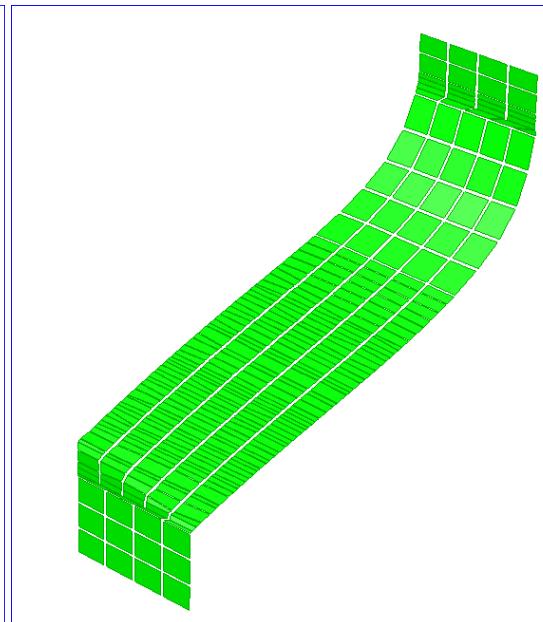
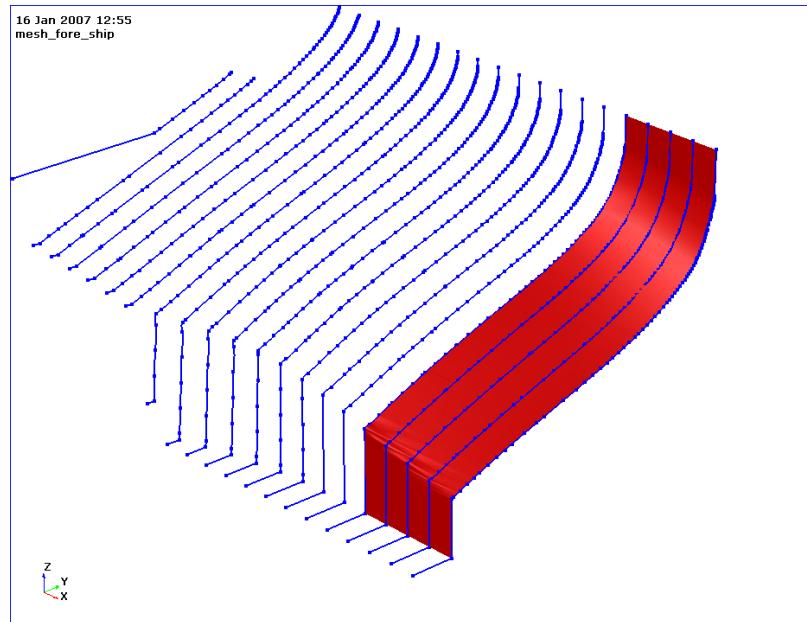
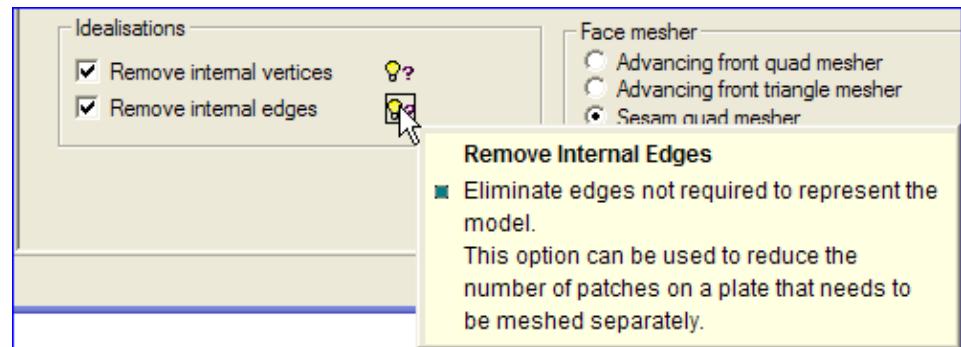
Revision date 25 Feb 2010



Slide 47

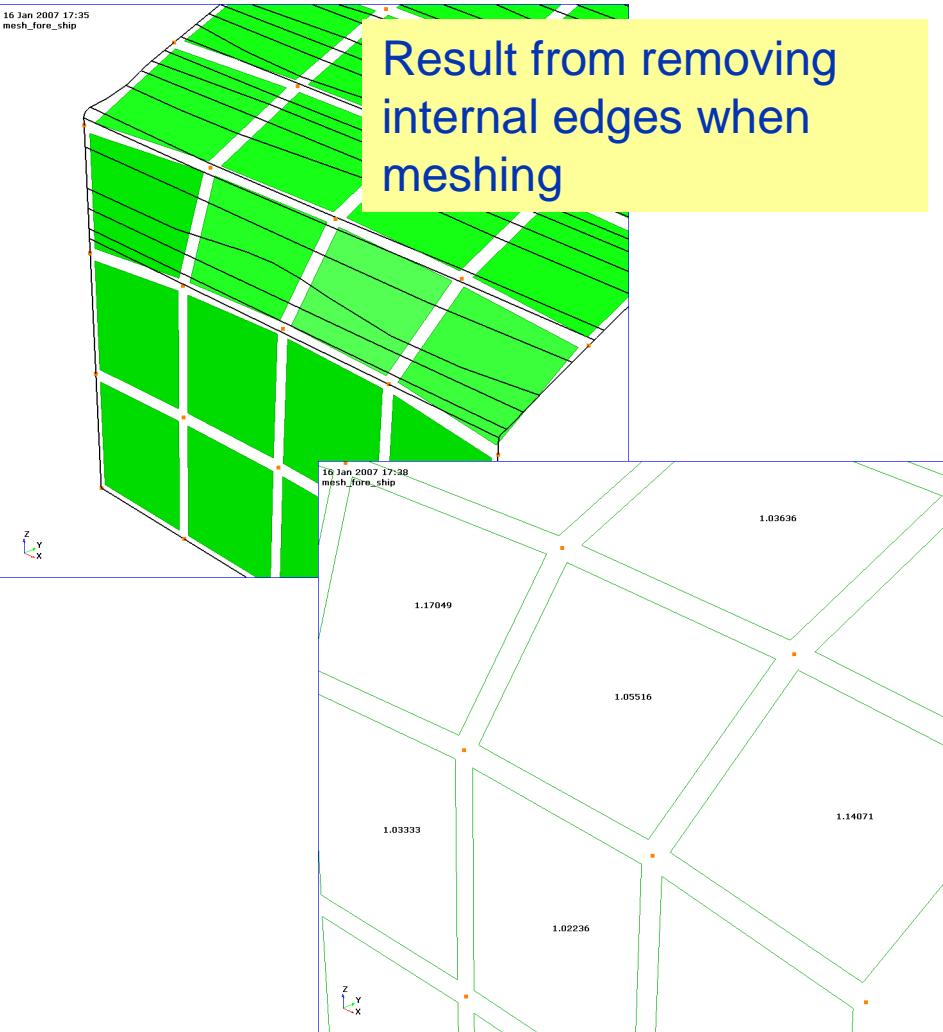
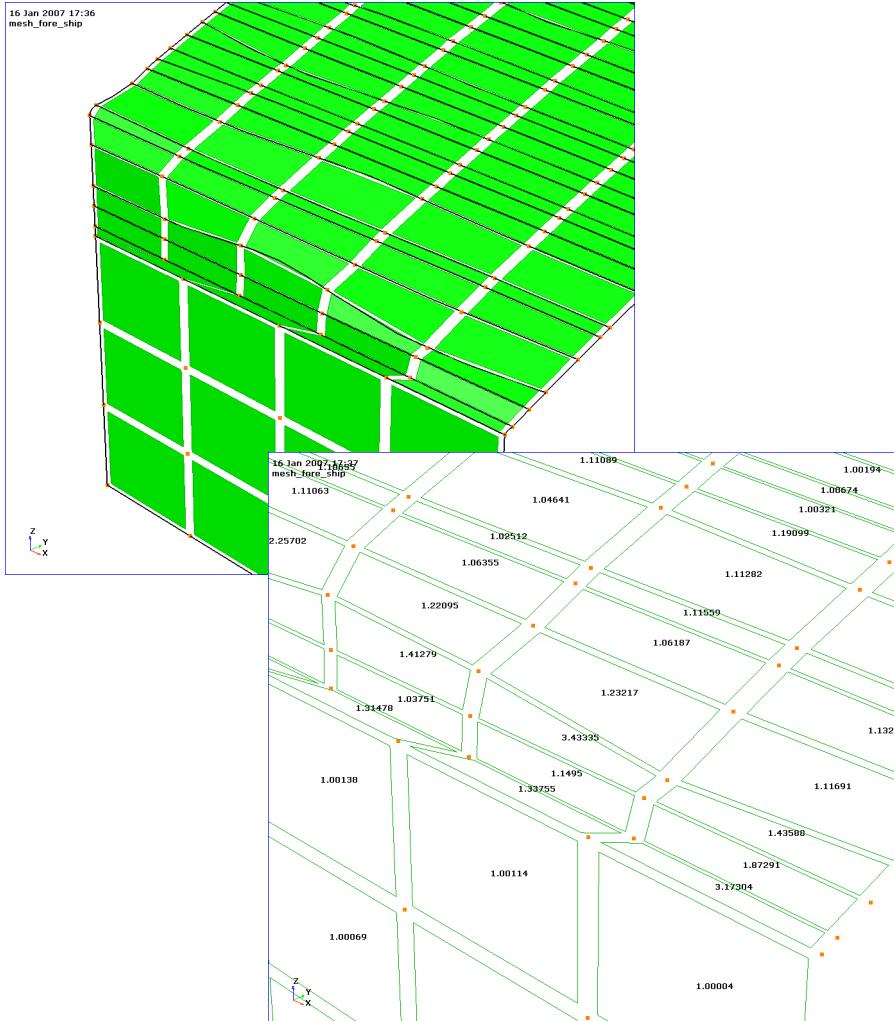
Remove internal edges

- Will in most cases reduce the number of patches on a plate that needs to be meshed separately
 - The default is: Disabled



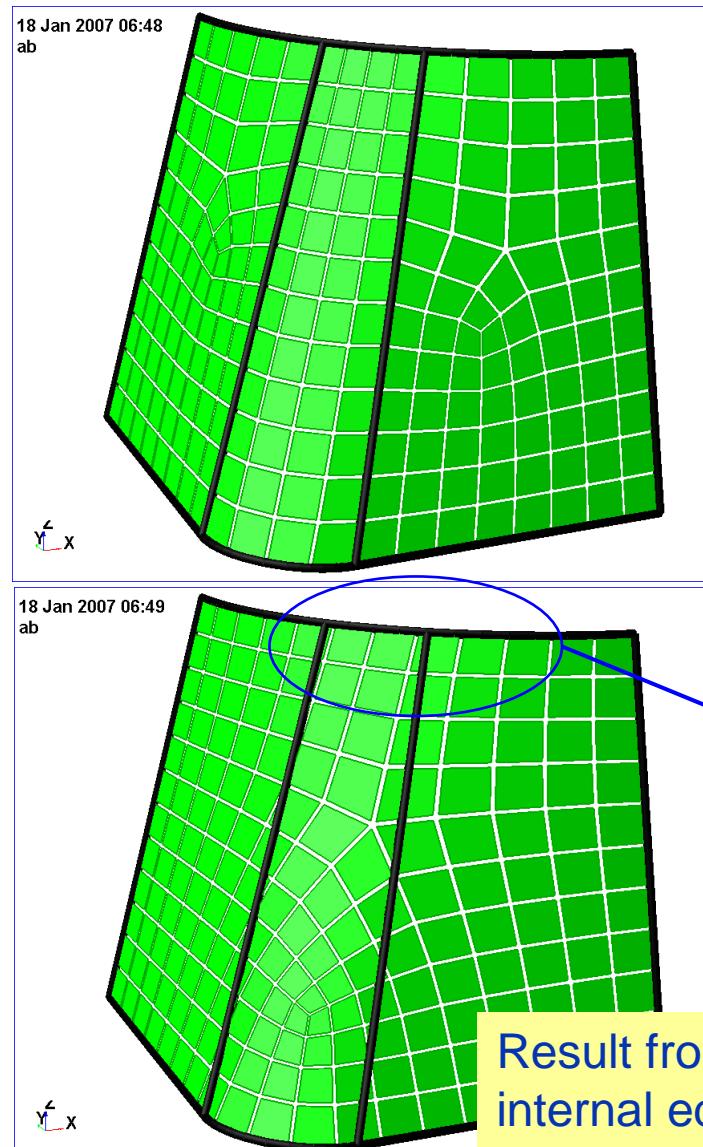
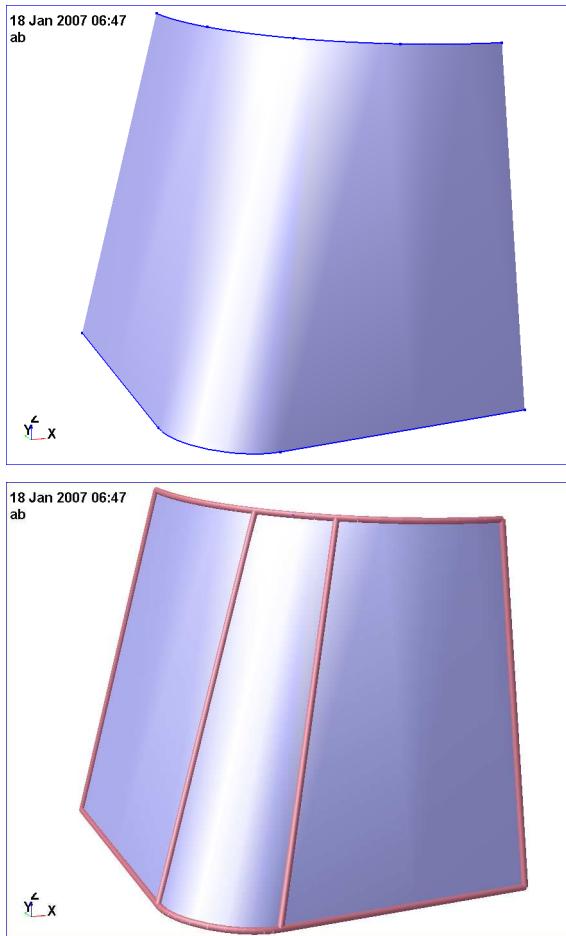
Remove internal edges

■ Behind the scene

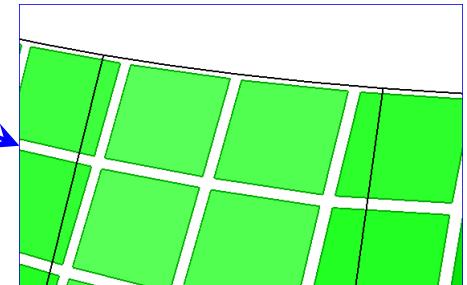


Remove internal edges

■ Another example



The mesh is now independent of internal edges



Result from removing internal edges when meshing

Other preferences

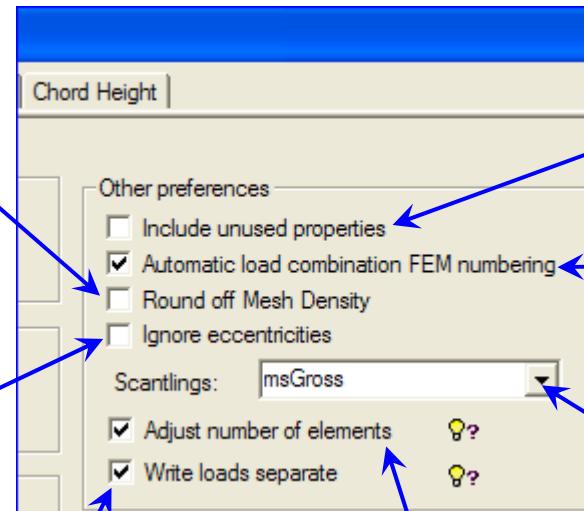
■ Special options

Allow the program some flexibility wrt. actual mesh size compared with characteristic mesh density

Ignore beam eccentricities when making the finite element mesh of the beams, i.e. aligned with the plate or neutral axis

Adjust number of elements

- Automatic adjust number of elements along edges should normally be enabled, as the face meshes normally will look better (with fewer triangles).
- Disable the option during problem solving if meshing hangs or if a poor mesh is observed, and check if this has any positive effect.

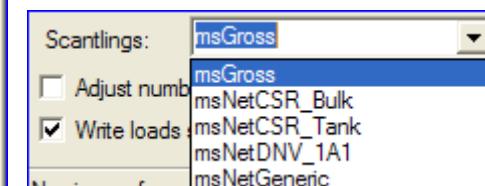


Include unused properties on the finite element file (FEM-file)

Let the program handle load combination numbering on the FEM file

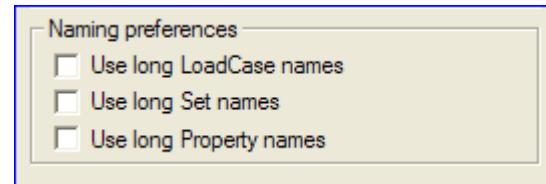
Adjust number of elements

- Automatic adjust number of elements along edges should normally be enabled, as the face meshes normally will look better (with fewer triangles).
- Disable the option during problem solving if meshing hangs or if a poor mesh is observed, and check if this has any positive effect.



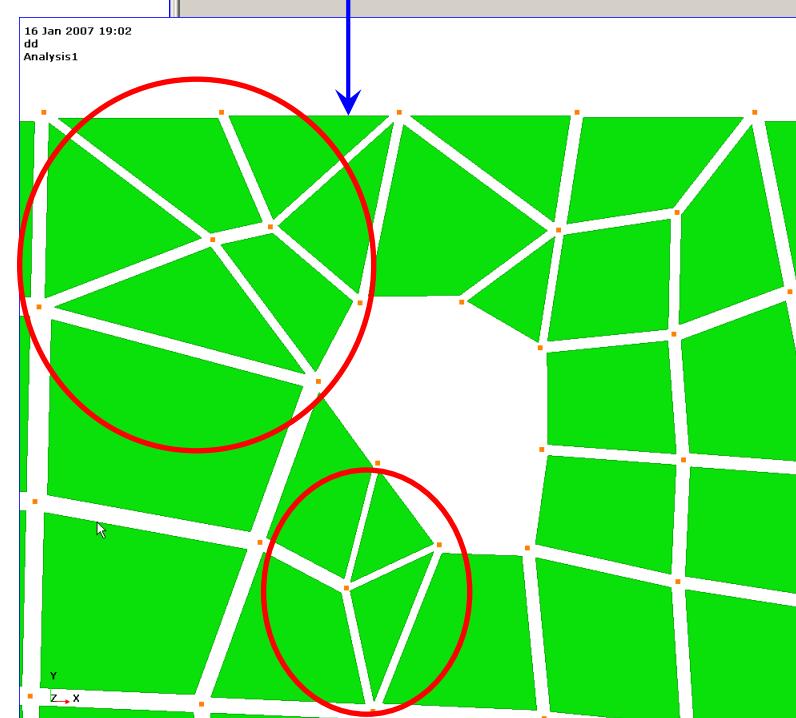
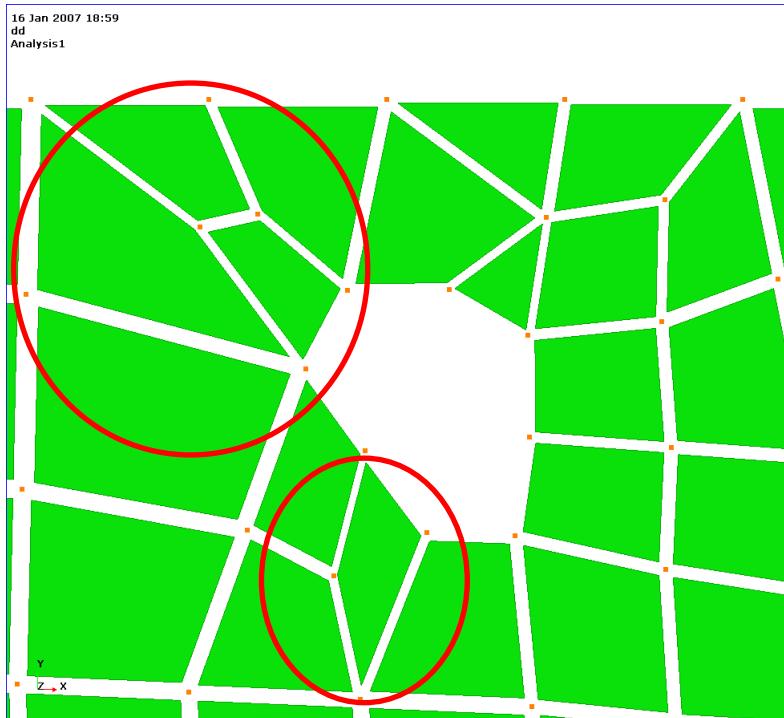
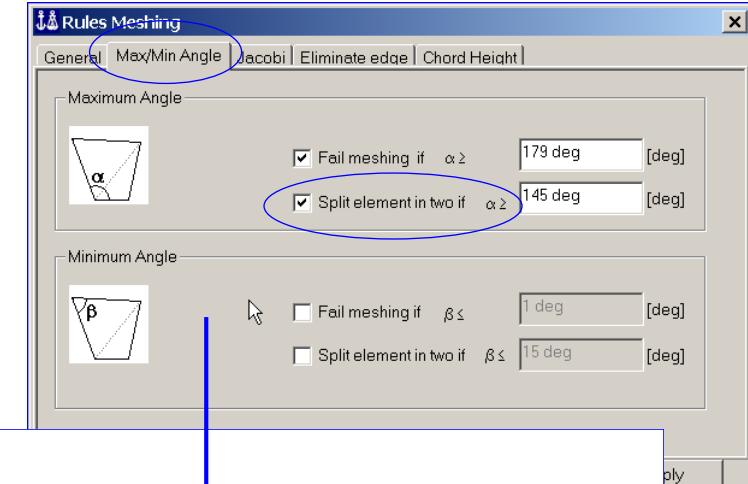
Naming preferences

- GeniE will write a FEM/SIN file that can be used by other programs
 - Xtract, Framework, Platework etc.
- Switching to “long” names enables the other programs to display names longer than 8 characters.



Max/min angle settings

- Program behaviour if criteria is exceeded

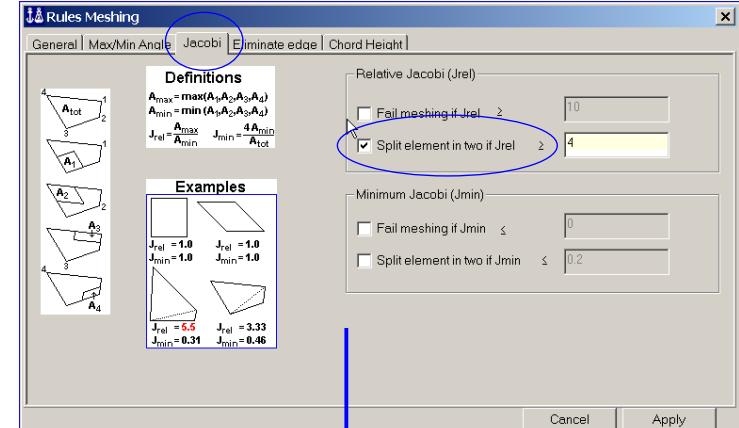


Jacobi max/min settings

- Program behaviour if criteria is exceeded
 - Notice: The Jacobi determinant > 4.0 is also listed in output from Sestra

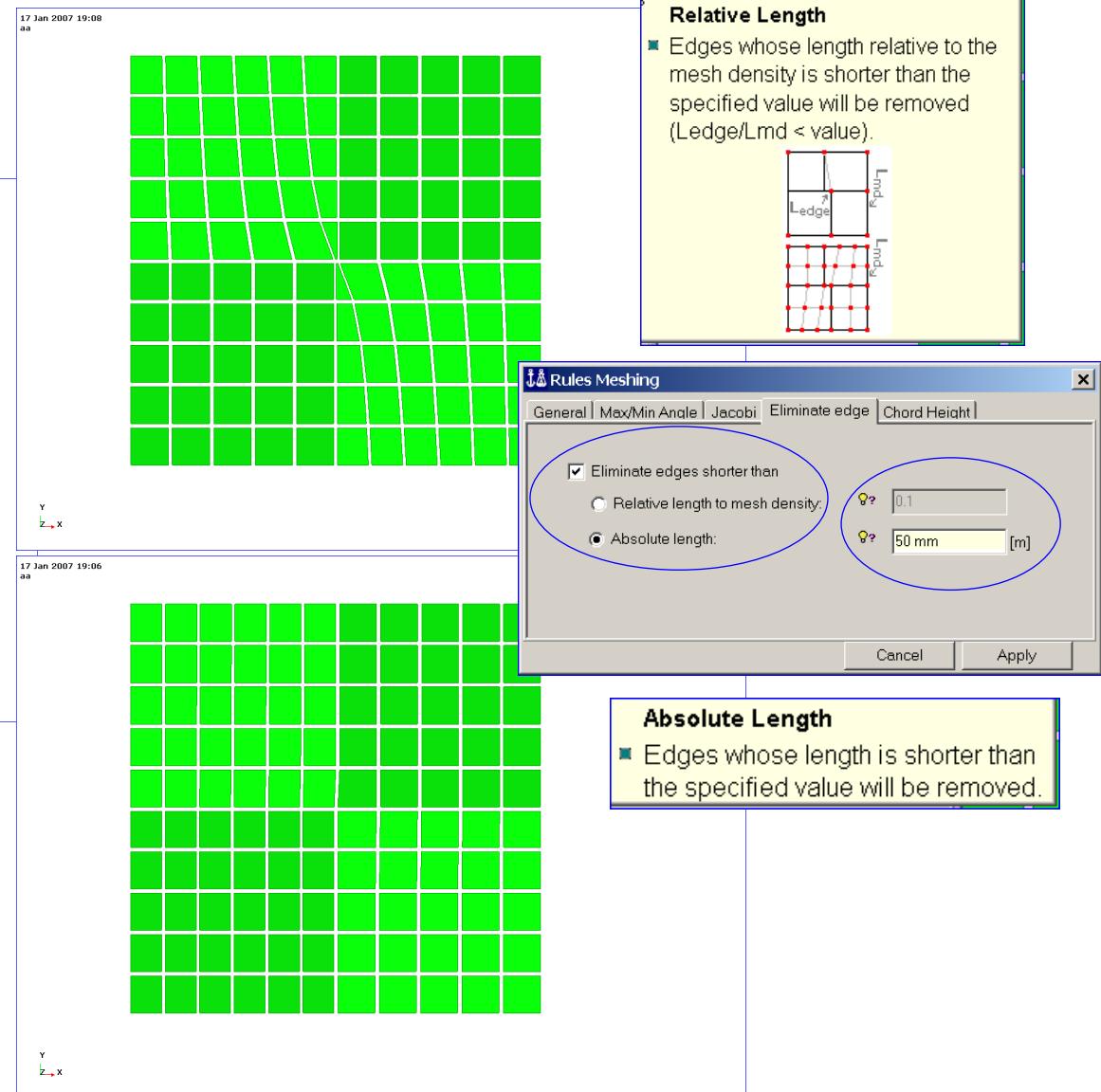
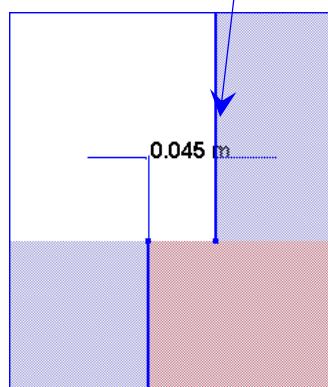
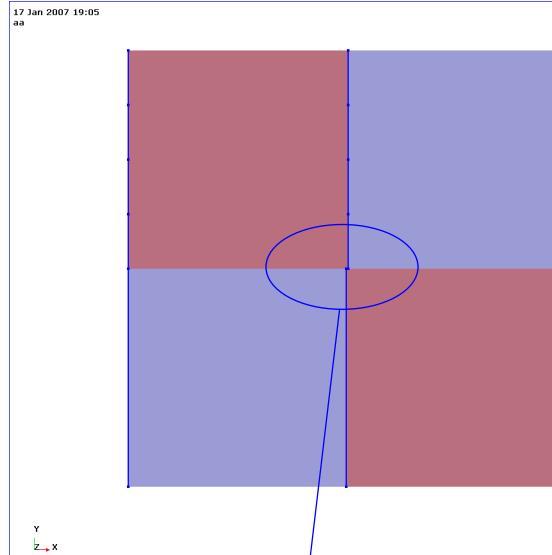
For a summary of warnings for elements and more information
see *** SUMMARY OF WARNINGS FROM DATACHECK OF ELEMENTS ***

```
*** WARNING Element no.      13 is distorted. The Jacobian Ratio is  5.2
*** WARNING Element no.      16 is distorted. The Jacobian Ratio is  5.2
*** WARNING Element no.      50 is distorted. The Jacobian Ratio is  4.9
```



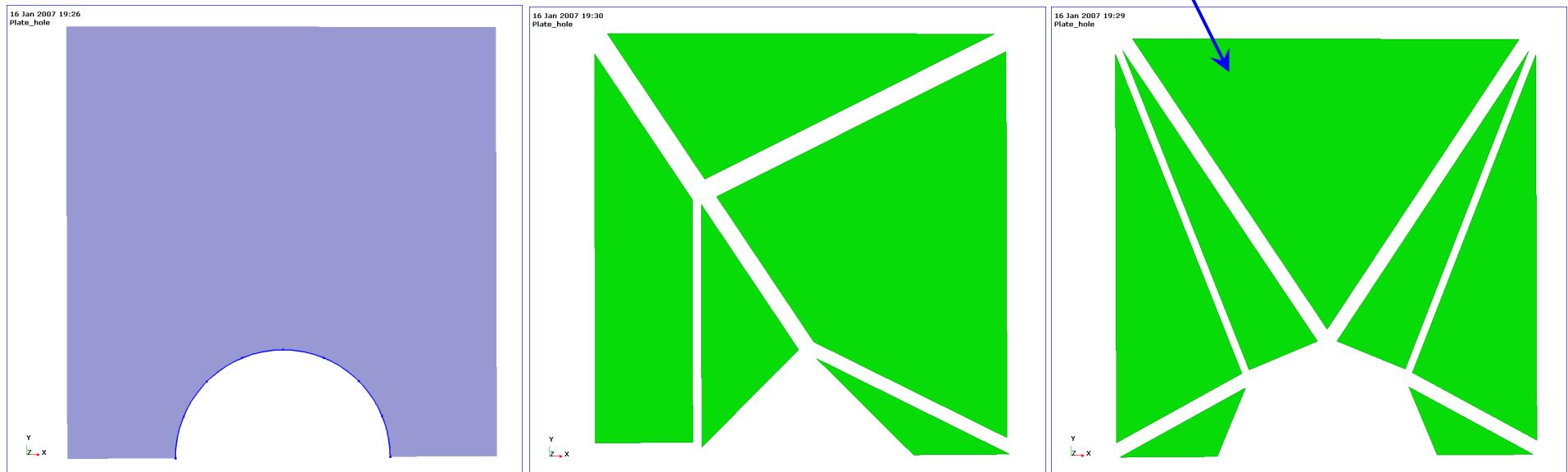
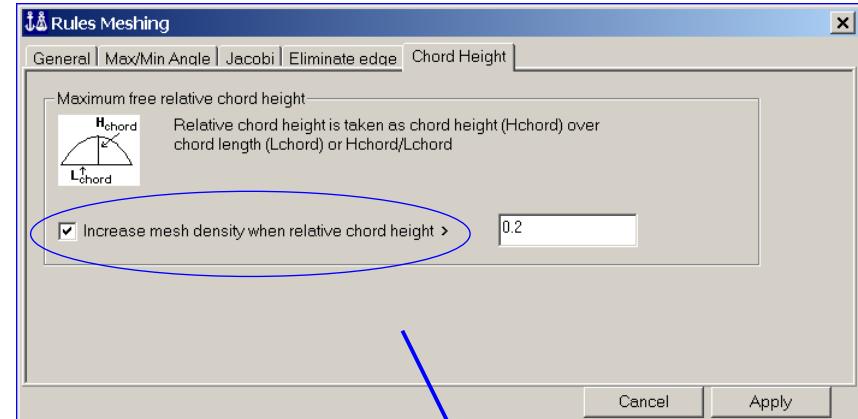
Global and eliminate edge

- Removing edges shorter than a given value



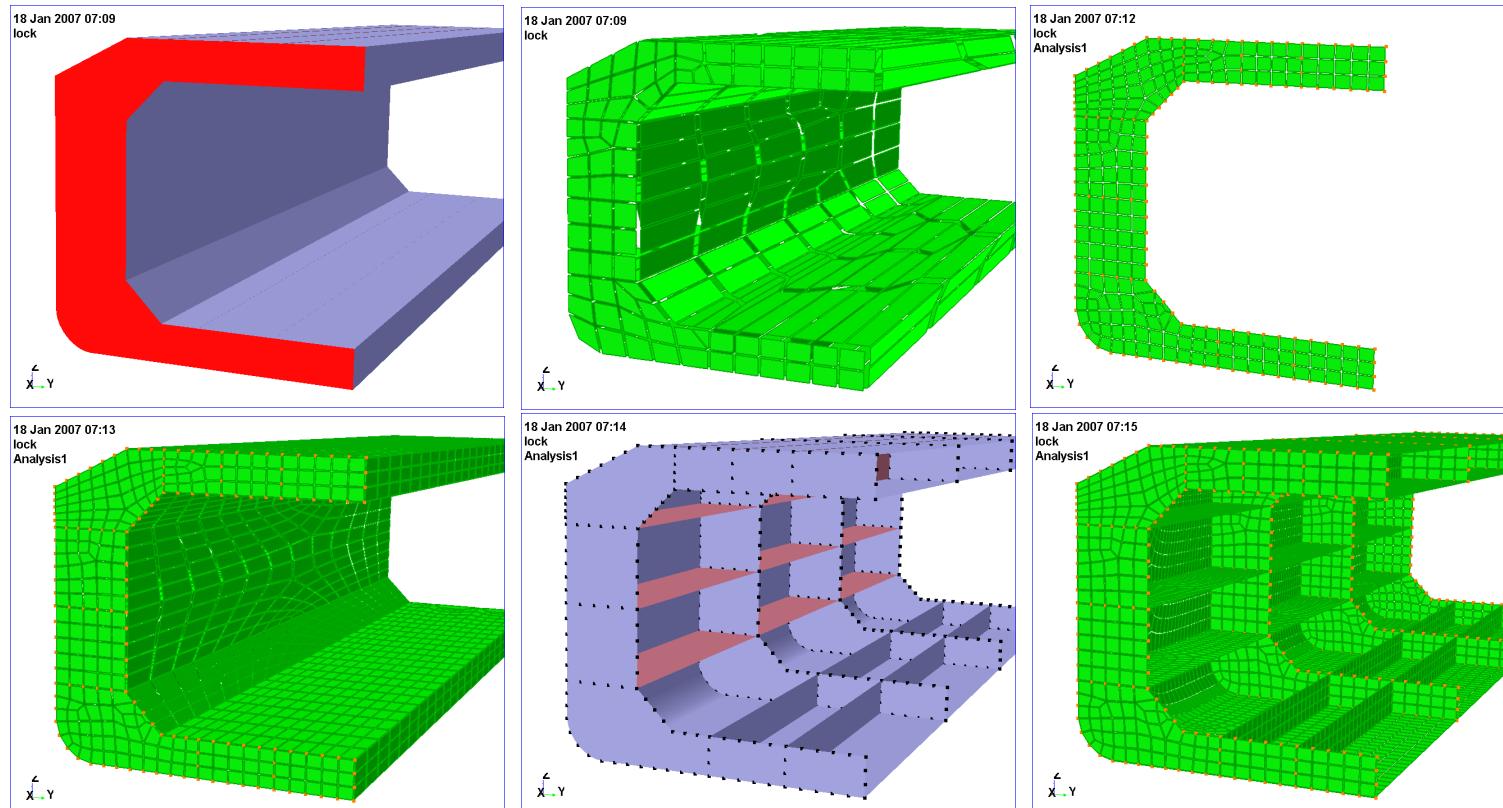
Chord height settings

- Mesh density is increased when setting a reference value for the relative chord height
 - This setting has normally an effect when global mesh setting is very coarse
 - In most cases paver meshing may be used instead



Mesh locking

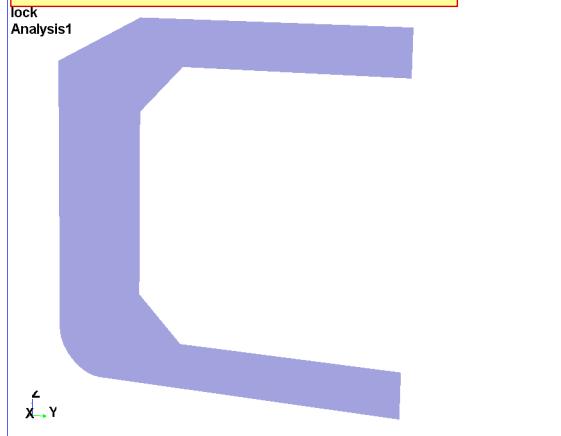
- It is possible to lock the mesh so that it remains constant independent of model modifications
 - Typically for a web-frame
 - The mesh locking is a property and will be part of a copy operation



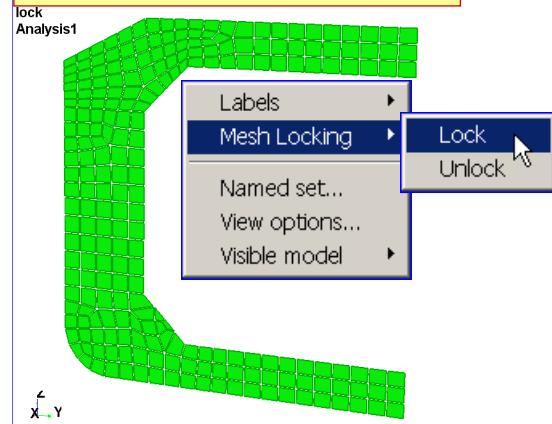
Mesh locking

■ How to do it?

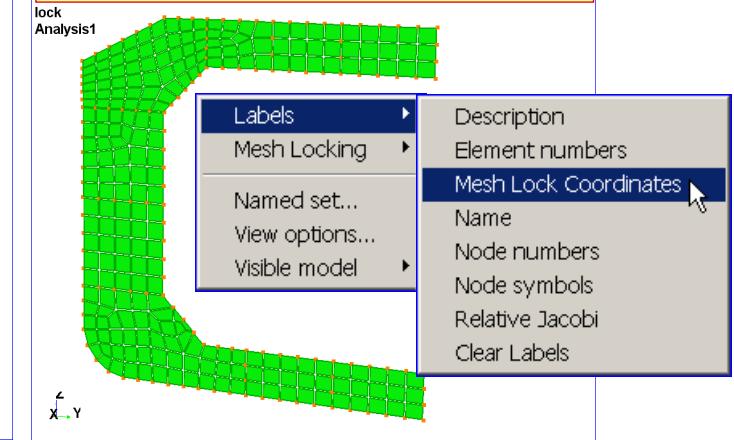
1. Select the part to focus and fine tune the mesh



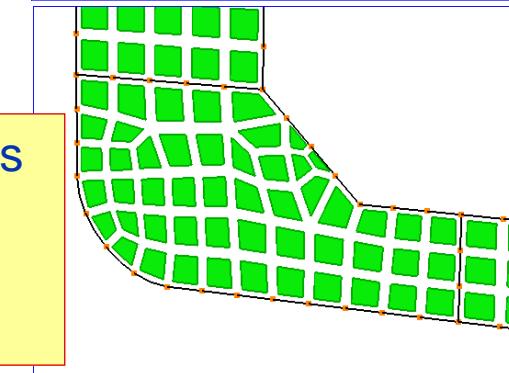
2. Select the part, right click and choose 'Lock'



3. Select the part, right click and choose 'Label Mesh Lock Coordinates'

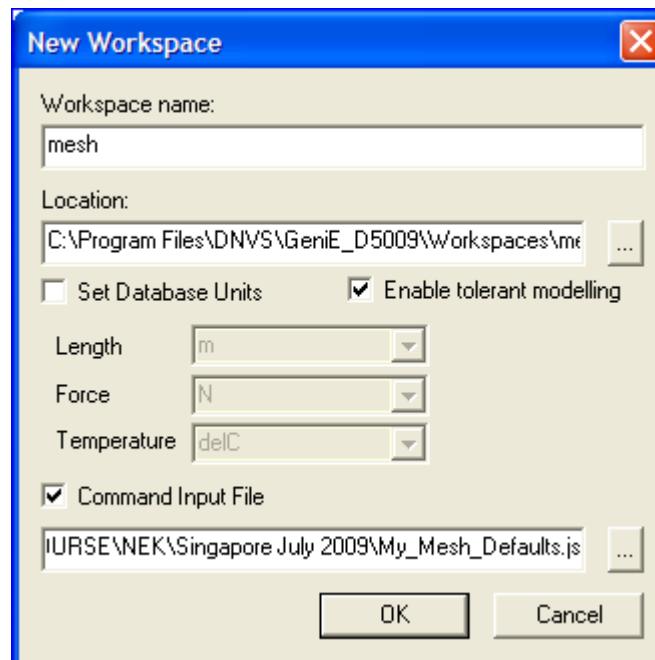


The mesh locking coordinates are inserted along topology edges and there may be a variation in-between



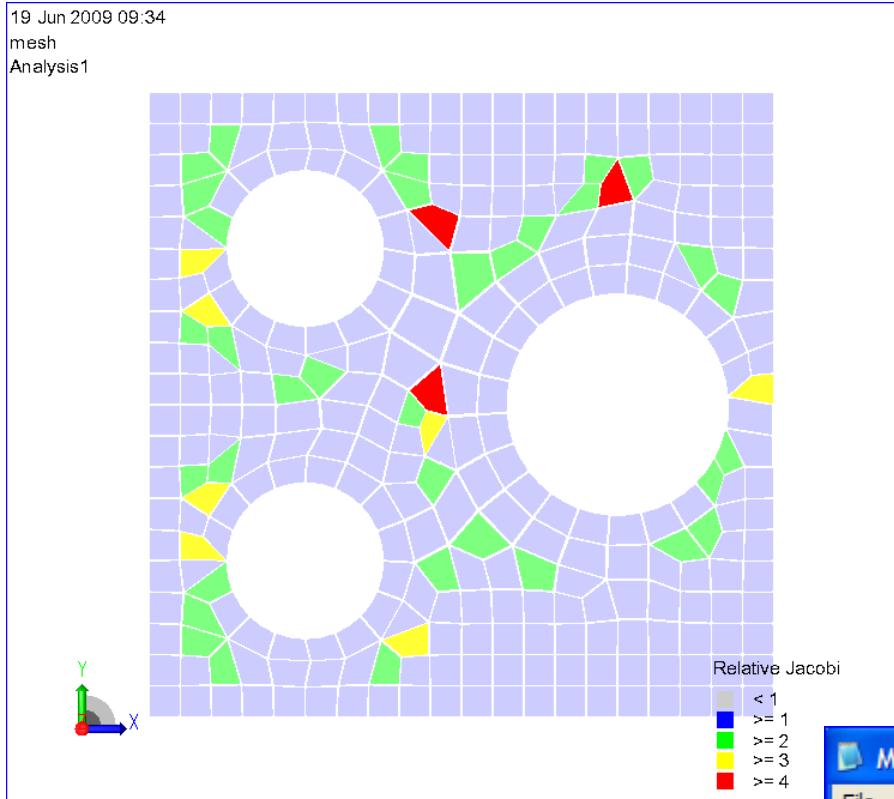
Make your own program defaults

- Your own mesh rules can be made program defaults
 - Create a js-file with your mesh settings (edit or from a graphical session)
- Refer to this file when making a new workspace
 - GeniE will always use this file when starting up until you de-activate the file



Make your own program defaults

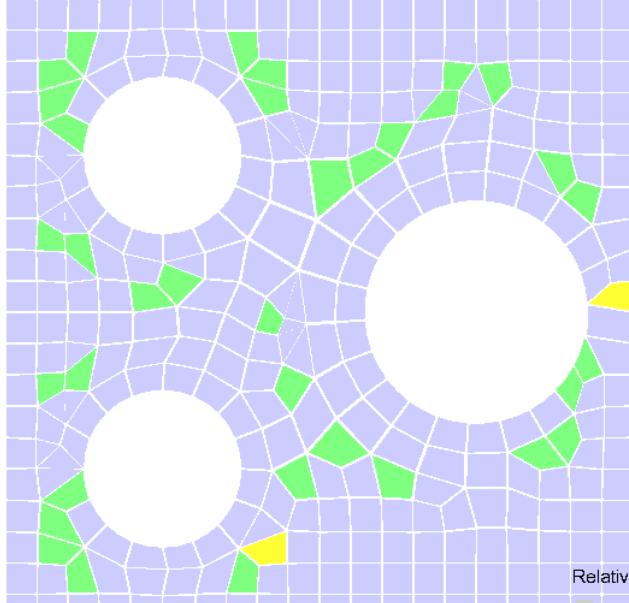
■ With program global defaults



Split in triangles if Jacobi > 4
or
angle > 145 degrees

■ With “your own” global defaults

19 Jun 2009 09:35
mesh
Analysis1



Relative Jacobi

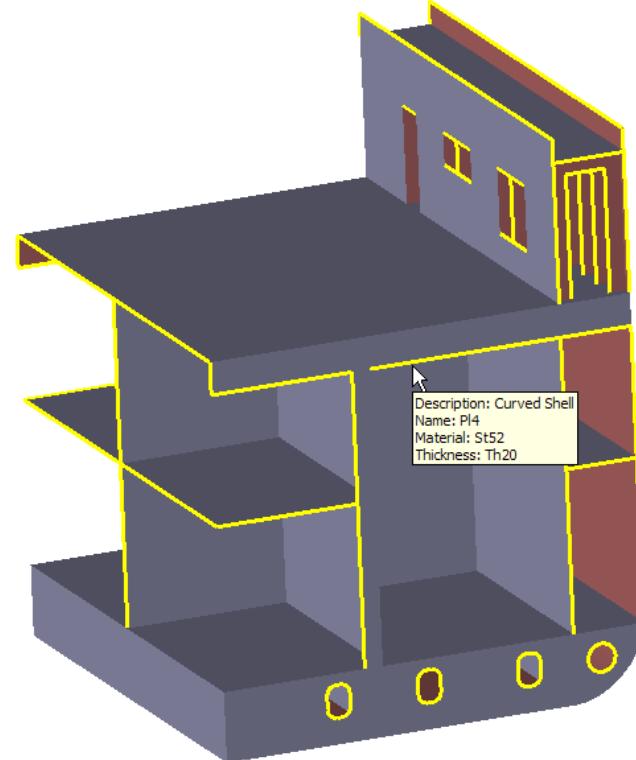
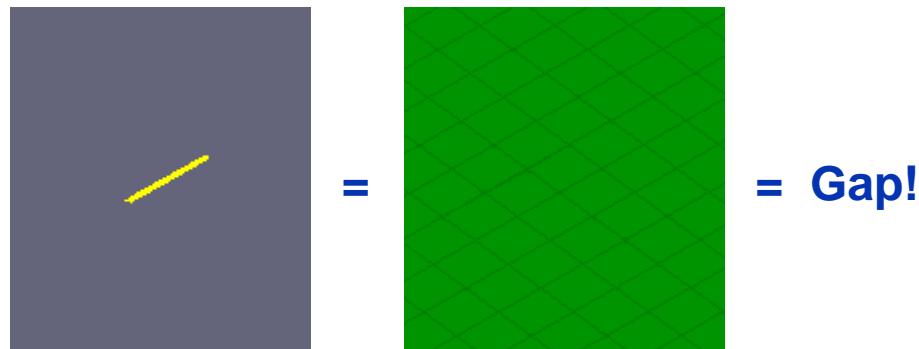
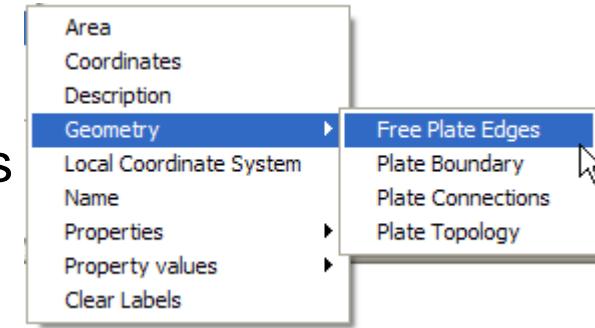
- < 1
- ≥ 1

My_Mesh_Defaults.js - Notepad

```
File Edit Format View Help
GenieRules.Meshing.elementType = mp2ndorder;
GenieRules.Meshing.faceMeshStrategy = AdvancingFrontQuadMesher;
GenieRules.Meshing.edgeMeshstrategy = LinearDistributionEdge;
GenieRules.Meshing.activate(mpMaxRelativeJacobi,mpssplit,true);
GenieRules.Meshing.setLimit(mpMaxRelativeJacobi,mpssplit,4);
GenieRules.Meshing.activate(mpMaxAngle,mpssplit,true);
GenieRules.Meshing.setLimit(mpMaxAngle,mpssplit,145);
```

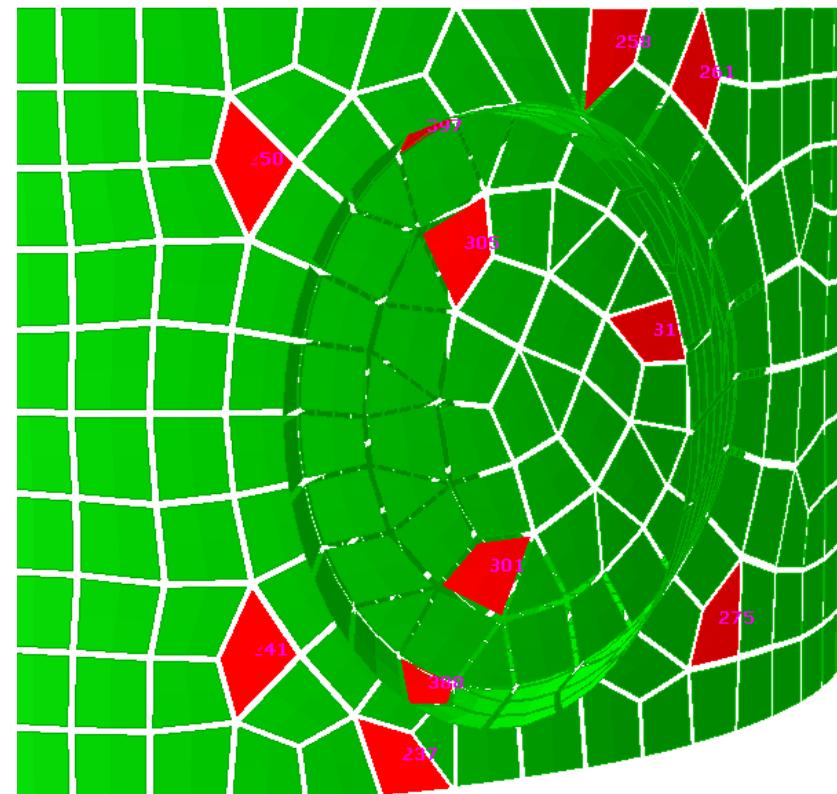
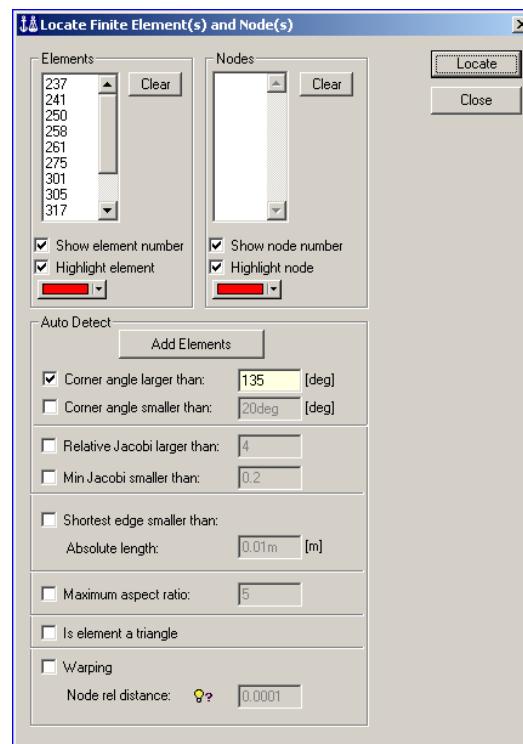
Free edges/boundary/connections/topology

- Labels | Geometry | Free Edges
- Free geometry edges => Free element edges
 - Nodes connected to only one element
- Free edges "in" model => Gaps
 - Even if all neighbouring elements seem OK



Locate FE

- Tools | Analysis | Locate FE
 - Find nodes & elements
 - Check element quality

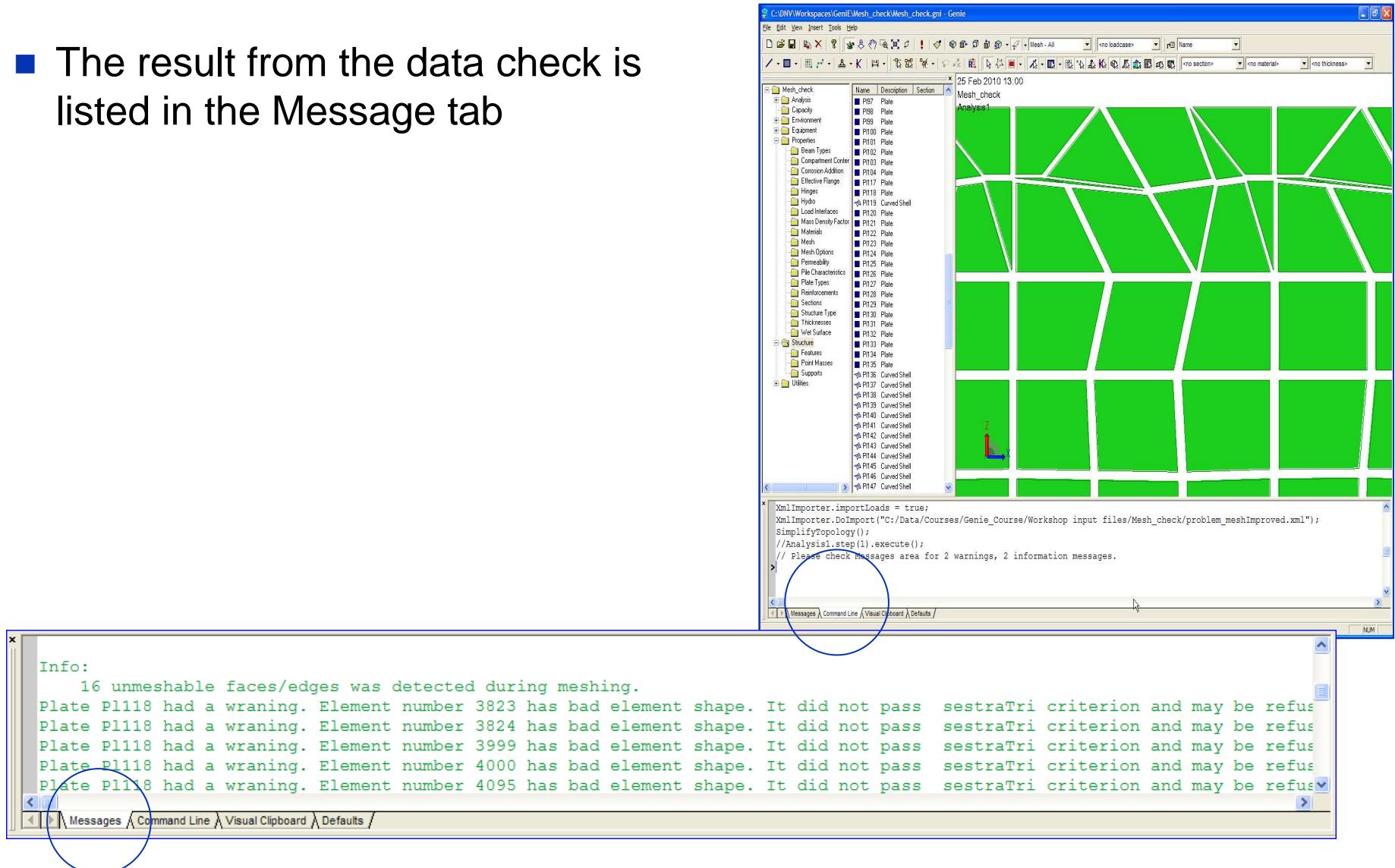


Data check of element quality

- GeniE will perform a data check of the elements prior to structural analysis (in Sestra)
 - The criteria used in Sestra for refusing elements are as follows:
 - For triangles:
$$100*A - L^2 < 0$$
Where A is area, L is length of longest edge
 - For rectangles:
$$25*C - D < 0$$
Where C is the shortest diagonal, D is the longest diagonal
- GeniE reports on elements that does not satisfy these criteria. It prints a warning message indicating element number and associated face, as well as the failing value.
 - See example on next page

Data check of element quality

- The result from the data check is listed in the Message tab





www.dnv.com
