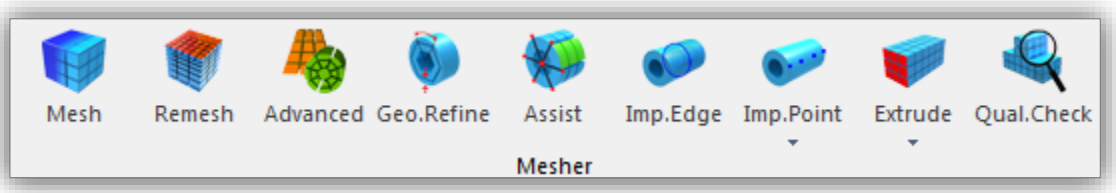
RECURDYN

V9R1 Mesher Enhancements and G-Manager Practice

FunctionBay

V9R1 Mesher Upgrade

❖ V9R1 Mesher



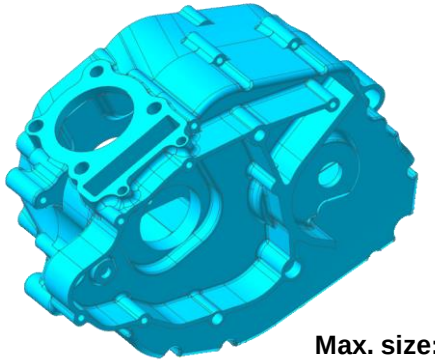
➢ Mesh Core Upgrade

① Mesh Core program is upgraded to the latest version.

② New mesher adapts the 64bit platform.

Therefore, user can get better mesh quality and more faster result by V9R1 NEW Mesher.

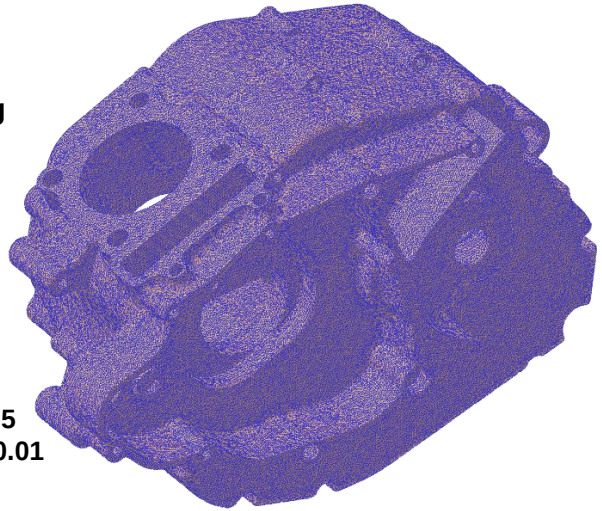
Complex CAD Geometry



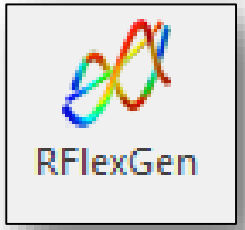
V9R1 Meshing

Max. size: 2
Min. size: 0.2
Gradation Factor: 0.5
Chordal Error Tol.: 0.01

Large Scale Flexible Body (1Million Nodes)



Generate the RFlexGen to get the large scale RFI file



V9R1 Mesher Upgrade

❖ V9R1 Improvement Points

1) Mesh Quality Improvements

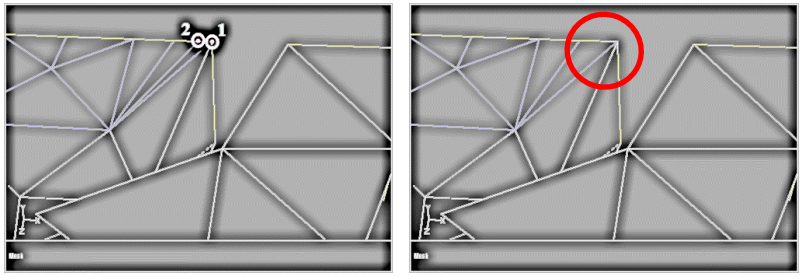
A. Edge and Node Merge algorithm Improvement → Unnecessary elements are not generated

B. Patch Angle Tolerance & Max. Chordal Error Option → User can control the surface tolerance

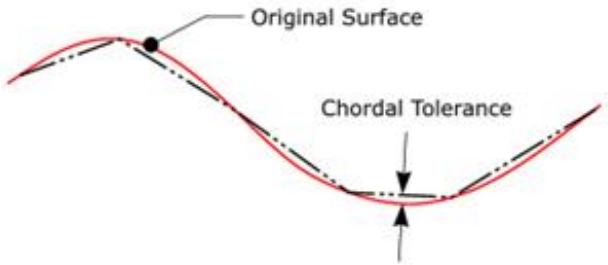
C. Mesh Gradation by controlling MIN/MAX size value → User can control the mesh gradation

D. Improvement of smooth surface mesh and min size → It can generate the smooth mesh surface by controlling the user defined MIN size value

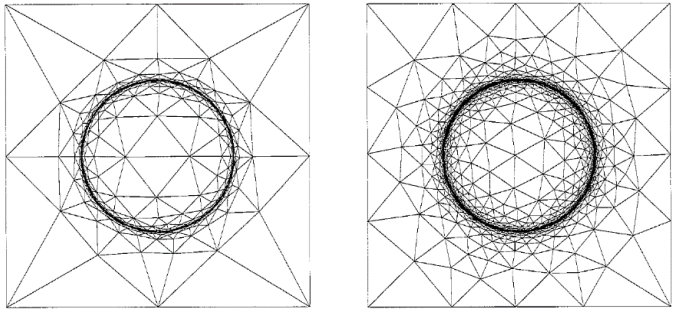
Edge and Node Merge Algorithm Improvement



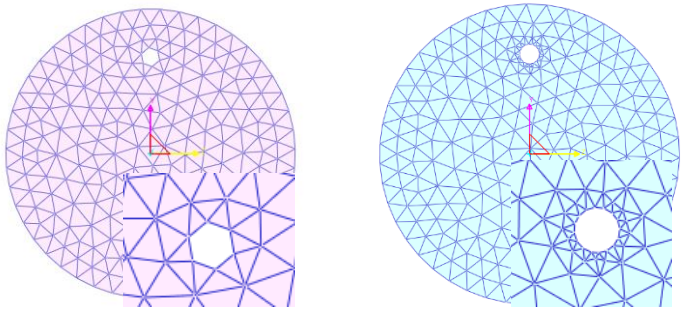
Max. Chordal Error Option



Mesh Gradation by controlling Min/Max size value



Smooth surface mesh by Min size value



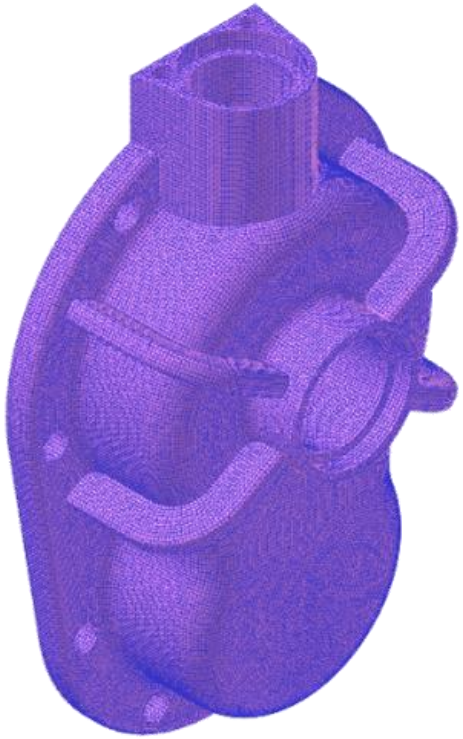
V9R1 Mesher Upgrade

2) Meshing Speed Improvement

A. Mesh core uses "Advancing Frontal Method" to the surface mesh, and then solid mesh is generated based on the surface mesh result. Then, it will make a improved solid mesh quality and generation speed.

B. Therefore, the number of nodes and elements will be less than previous mesher.

C. "**Hexa mesh**" has been improved in meshing speed about **2.5 times** faster. In the case of "**Tetra mesh**", its meshing speed has been improved about **1.4 times** faster.



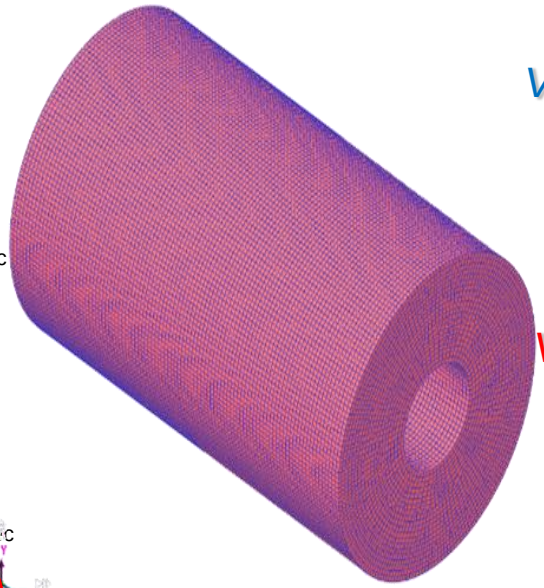
V8R5

Input
Min **4.676e-02**
Max **4.676e-02**
Output
Node **252655**
Element **321510**
Quality **111.251**
Time **486.292** sec

V9R1

Input
Min **4.676e-02**
Max **4.676e-02**
Output
Node **255720**
Element **313843**
Quality **16.671**
Time **184.881** sec
Speed **+62%**

2.5x Faster!



V8R5

Input
Min **10**
Max **10**
Output
Node **257065**
Element **266630**
Quality **11.168**
Time **291.193** sec

V9R1

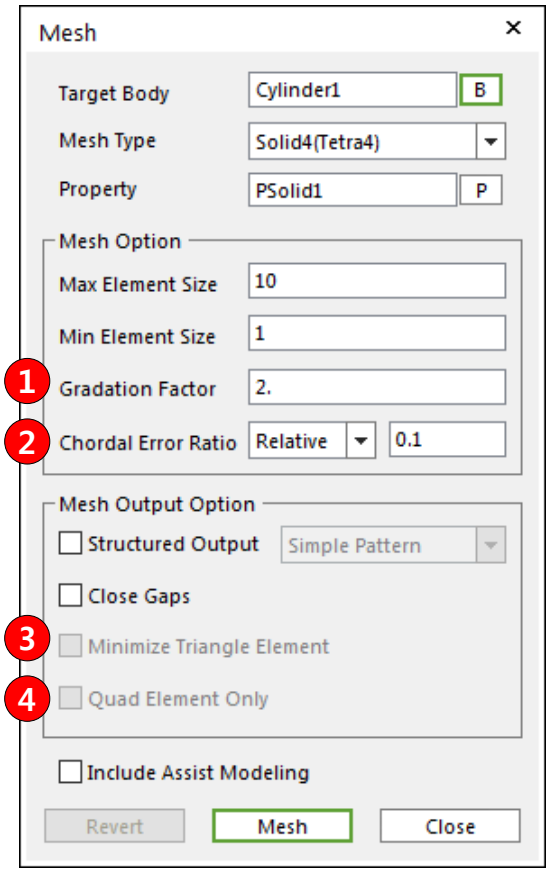
Input
Min **10**
Max **10**
Output
Node **248402**
Element **277284**
Quality **34.033**
Time **121.064** sec
Speed **+59%**

1.4x Faster!

Mesh (Auto Mesh) Upgrade

❖ Mesh Upgrade

: The core of the mesher was upgraded and new mesh options were developed. These new mesh options can give you more improved meshing quality results. In addition, user can control the meshing result using these new options.



New Options

1. **Gradation Factor**
   1) This gradation factor option can control the gradation of the element size from the boundary sizes to the defined size.
   2) A value close to 0 leads to a more progressive variation of mesh size.

2. **Chordal error ratio**
   1) The mesh size is reduced locally to limit the chordal error between the mesh and the geometry surface.
   2) If you select ‘Relative’ option, the value is percentage of the local radius.
   3) If you select ‘Absolute’ option, the value is absolute tolerance value.
   4) Default = Relative & 0.1 (10% of local radius)

3. **Minimize triangle element**
   1) Available for solid8 (hexa), shell4 (quad) mesh types.
   2) If this option is used, mesher generates the minimum number of triangle elements

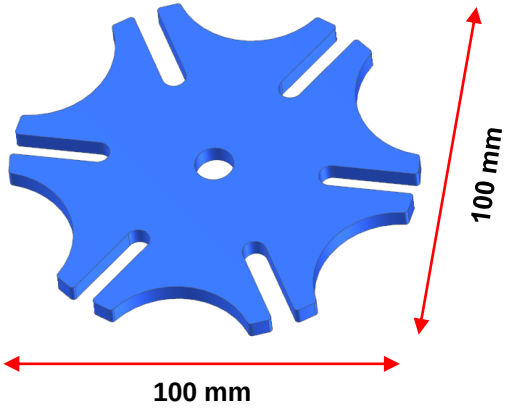
4. **Quad element only**
   1) Available for shell4 (quad) element type.
   2) If this option is used, mesher generates only quad elements.

Mesh Upgrade (Gradation Factor)

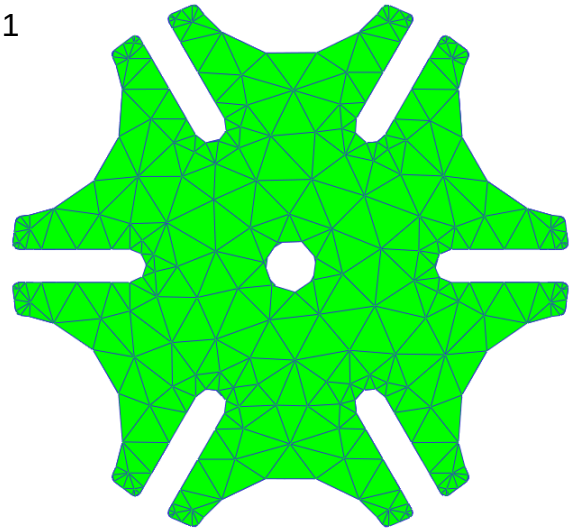
➢ How to use the new options

◆ **Gradation Factor**

- A value close to '0' leads to a more progressive variation of mesh size,
- The default value, '2' leads to a similar mesh result with previous V8 mesh result.

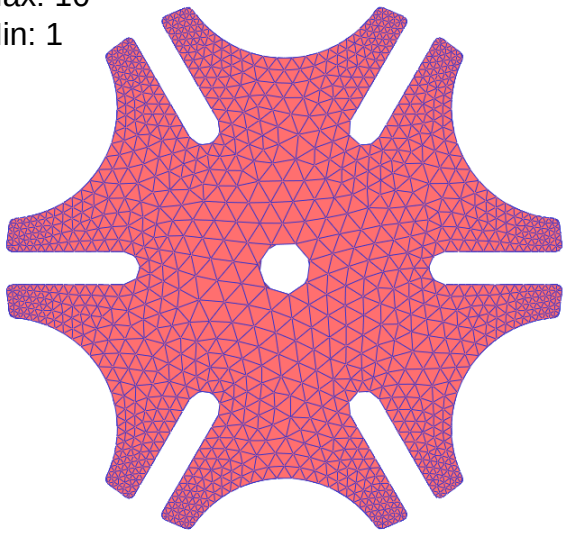


- **Gradation Factor: 2**
- Max: 10
- Min: 1



▶ ***Gradation Factor "2" leads a loose min-size-elements gradation effect.***

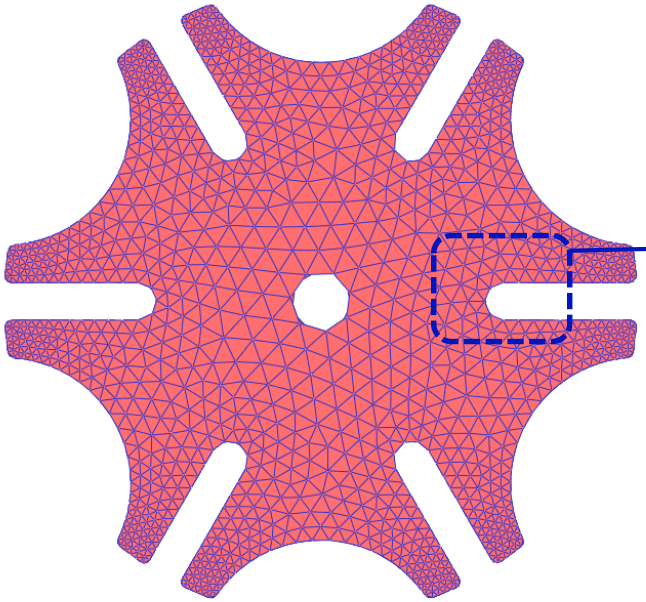
- **Gradation Factor: 0.1**
- Max: 10
- Min: 1



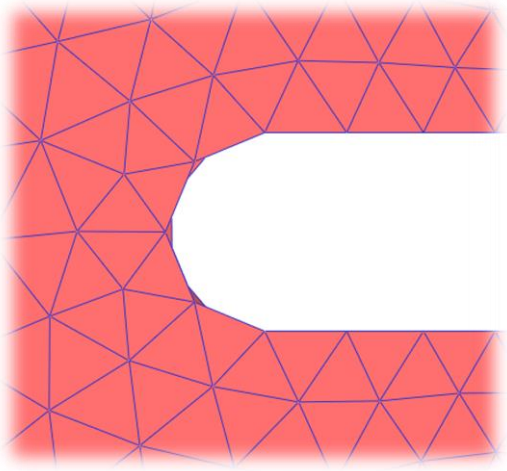
▶ ***Gradation Factor "0.1" leads an progressive min-size-elements gradation effect.***

Mesh Upgrade (Chordal Effect Ratio)

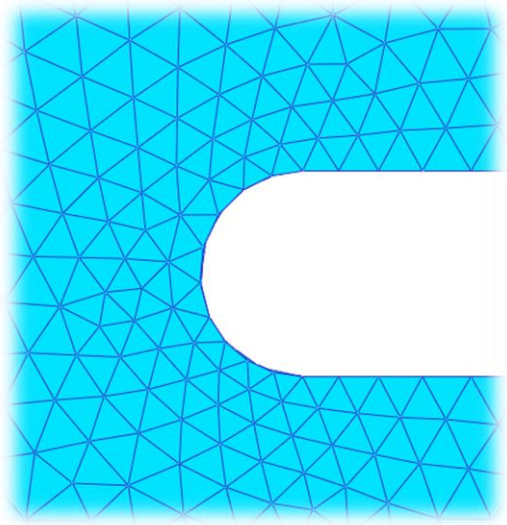
- How to use the new options
  - **Chordal Effect Ratio**
    - The mesh size is reduced locally to limit the chordal error between the mesh and the geometry surface.
    - If you select 'Relative' option, the value is percentage of the local radius.
    - If you select 'Absolute' option, the value is absolute tolerance value.
    - Default = Relative & 0.1 (i.e. 10% of local radius)



- **Chordal Effect Ratio: Relative, 0.1**



- **Chordal Effect Ratio: Relative, 0.01**



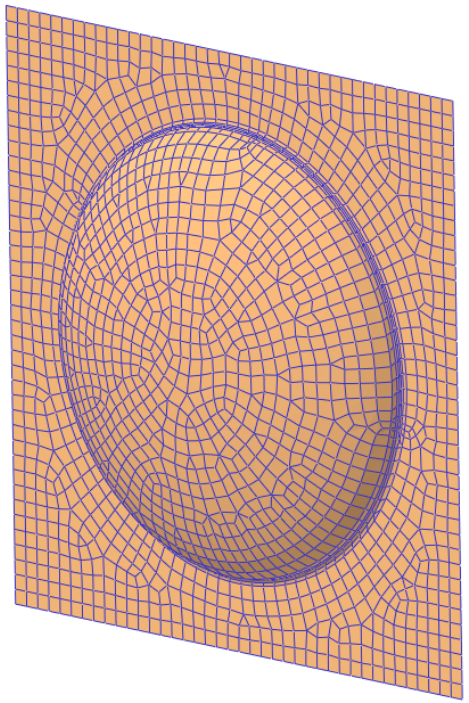
Mesh Upgrade

- How to use the new options
  - **Minimize Triangle Element & Quad Elements Only**

    If 'Minimize Triangle Element' option is used, mesher generates the minimum number of triangle elements

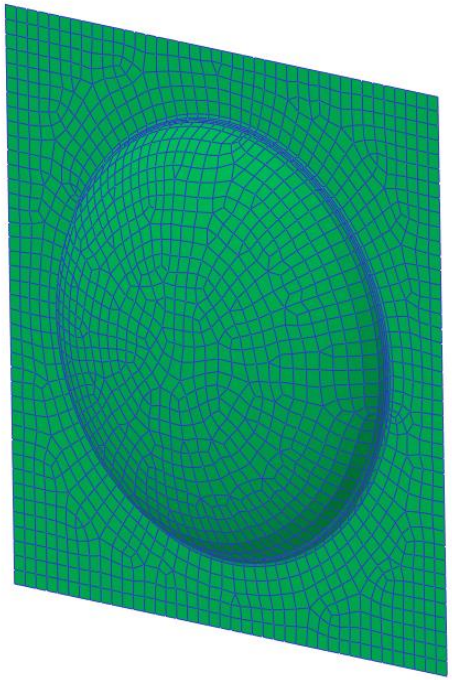
    If 'Quad Elements Only' option is used, mesher generates only quad elements.

- **No Options(Default)**



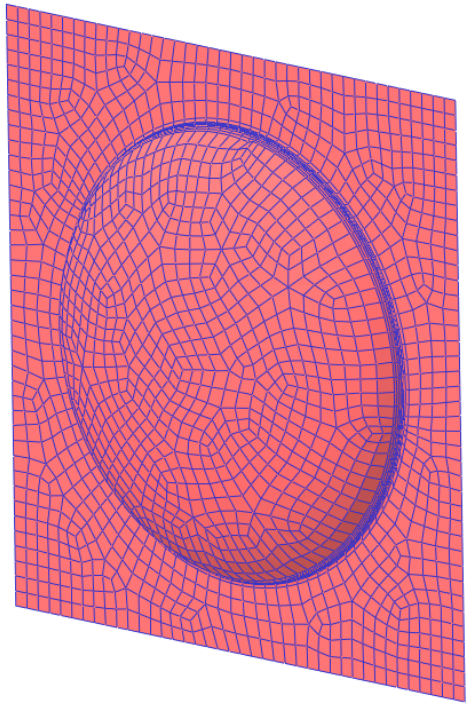
Shell4: 2087
Shell3: 92

- **Minimum Triangle Element: ON**



Shell4: 2178
Shell3: 62

- **Quad Elements Only: ON**



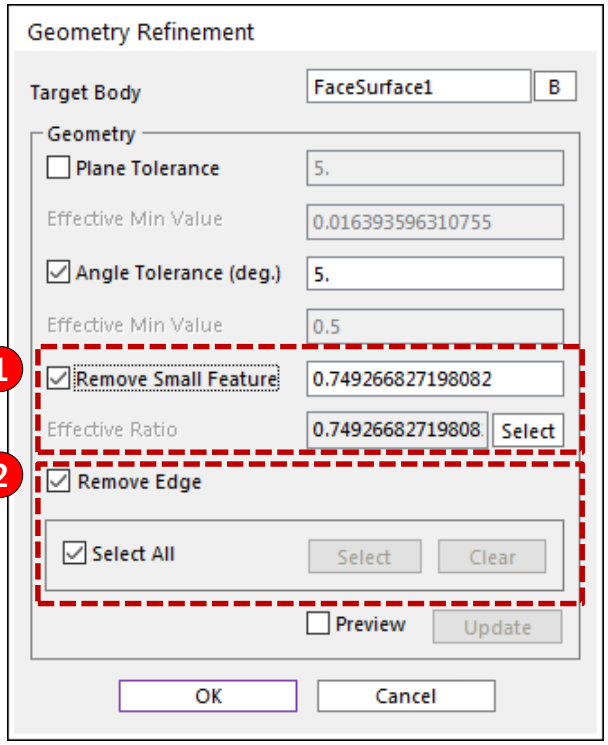
Shell4: 2548
Shell3: 0

Geometry Refinement

Geo.Refine

❖ Geometry Refinement

: Using the "Geometry Refinement" functions, user can re-generate the geometry data for meshing. And also, using the new Remove Edge function, user can control the geometry for meshing whether edge is used or not while meshing operation.

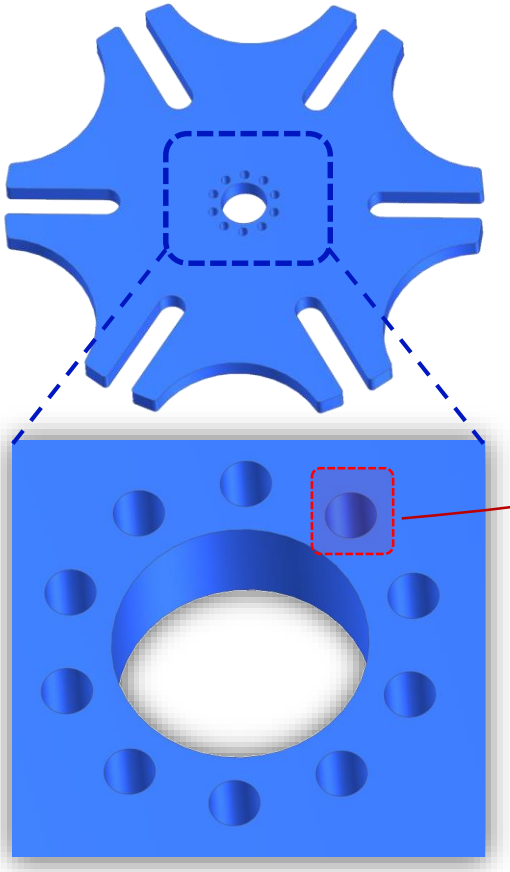


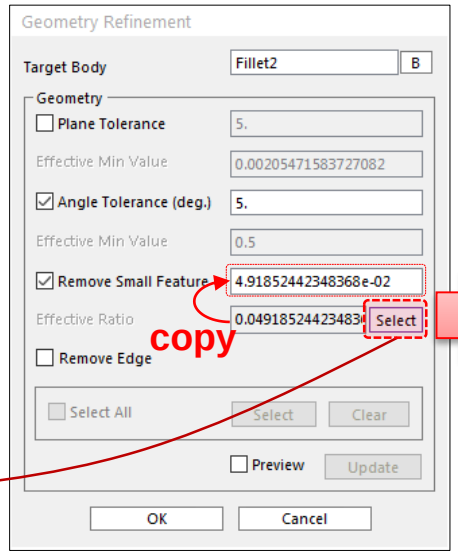
1. **Remove Small Feature**
   1) User can remove small features of the original geometry
   2) **Effective Ratio (enhanced option)** is calculated from the bounding box of the selected geometry
   3) User can select the target geometry which needs to be removed using 'select' button.

2. **Remove Edge Option (New option)**
   1) This option allows user to simplify a solid and surface geometry by removing the selected edges.

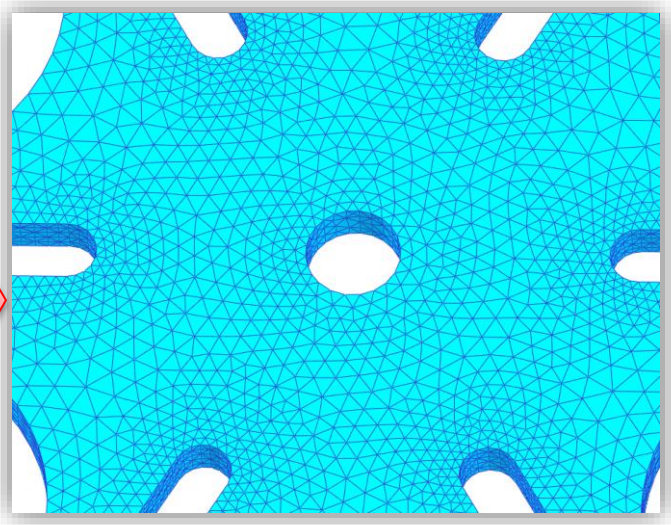
Geometry Refinement

- How to use the new options
  - **Remove Small Feature – Effective Ratio**
    - Mesher ignores the small geometries from the original geometry. (based on 'Effective ratio)
    - User can get the recommended 'effective ratio' using the 'Select' button. (after then, the recommended value needs to be input manually.)

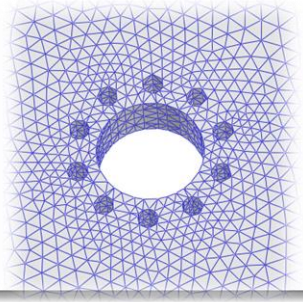




- **Remove Small Feature: ON**



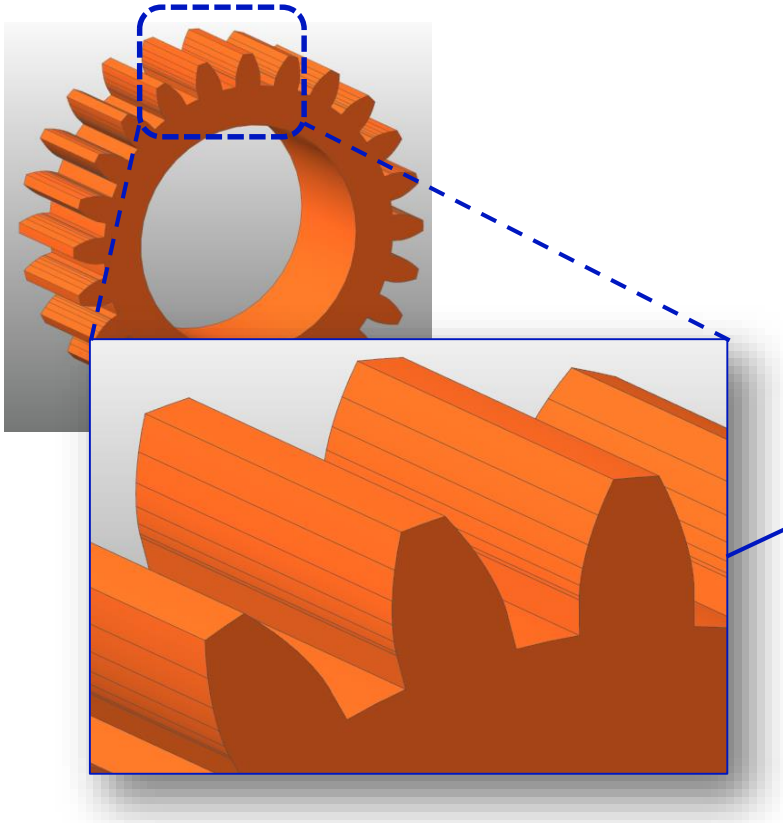
- **Remove Small Feature: OFF**



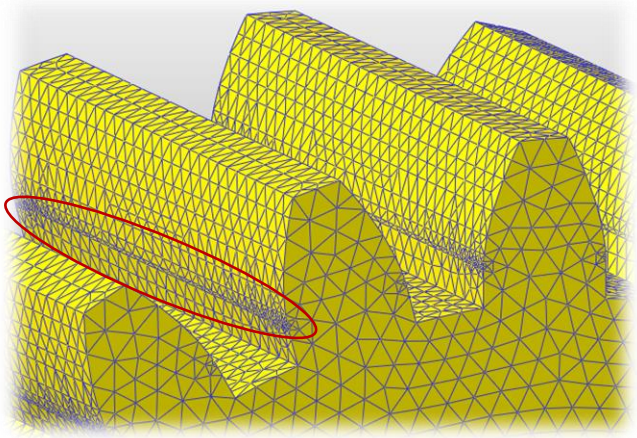
Geometry Refinement

- How to use the new options
  - **Remove Edge**

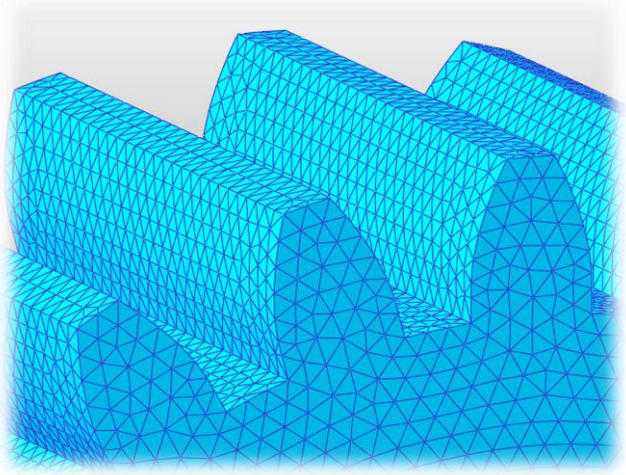
    - If 'Remove Edge' is checked on, mesher ignores the selected edges.



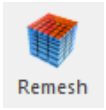
- **Remove Edge: Check Off**



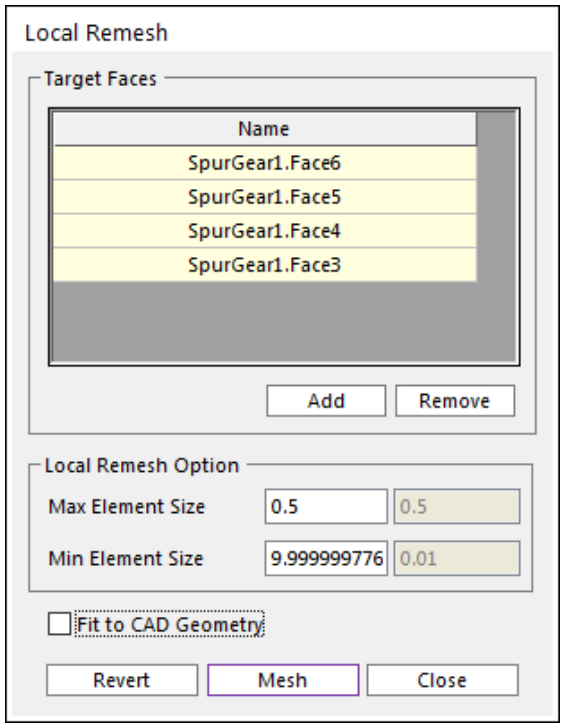
- **Remove Edge: Check On(Select All)**



Local Remesh



❖ Local Remesh – Multiple Local Remesh



A. Multiple re-mesh in Local Remesh
: User can apply local Remesh several times on the different surface

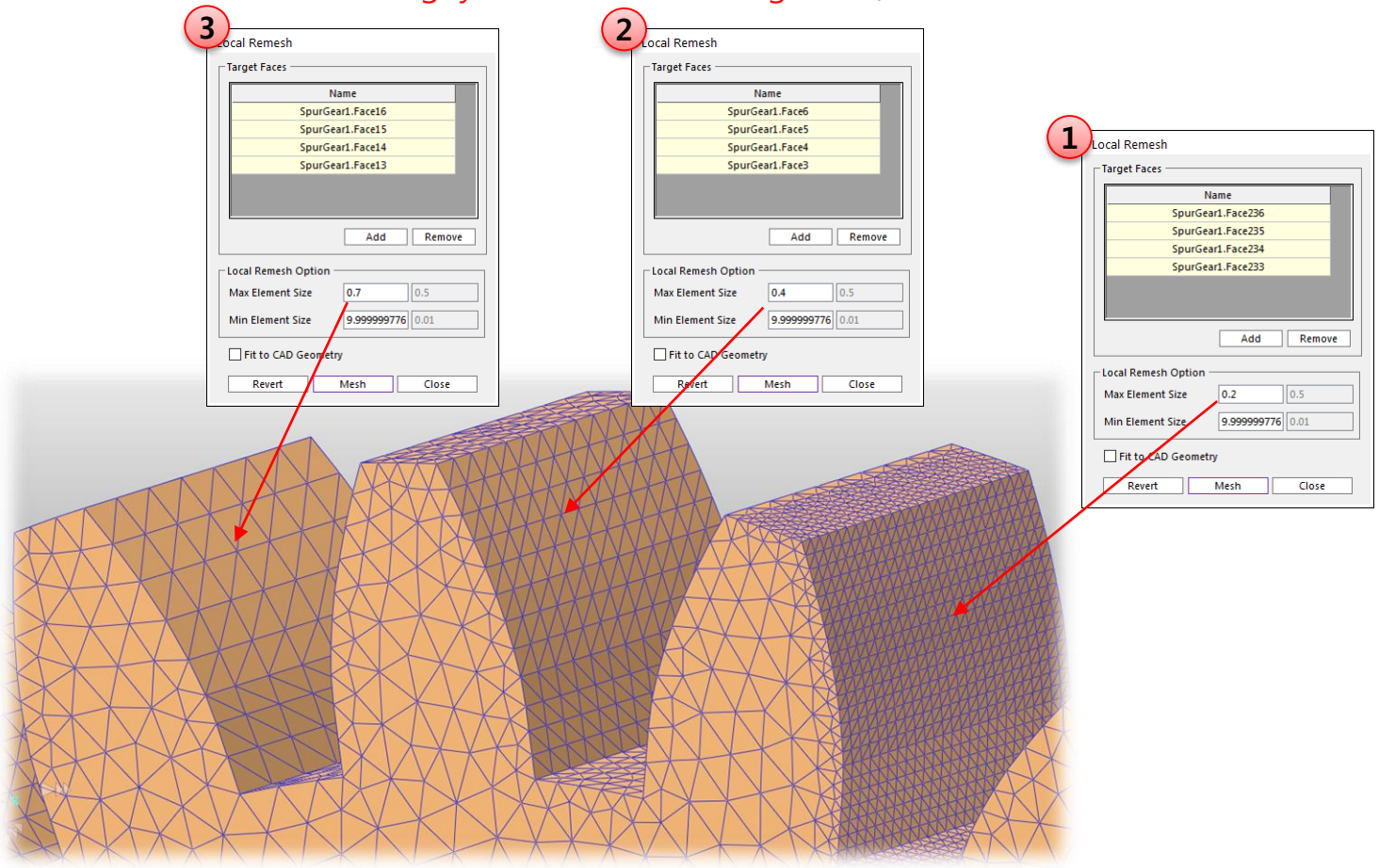
※ Note: if you check "Fit to CAD Geometry" option, RD does NOT keep the previous local remesh result. It means RD does NOT support multiple remesh function with "Fit to CAD Geometry".

Local Remesh

- How to use the new options
  - **Local Remesh & Advanced mesh**

    : After auto-meshing or advanced meshing, user can define the different Remesh settings on the different surfaces. For example, user run the first remesh on a surface, and then **close the remesh dialog.** And, **open again** the remesh dialog and run another remesh and close it.

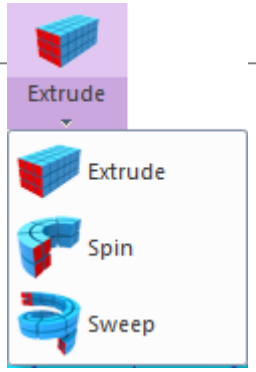
    (To apply the different Remesh setting, you should the dialog once.)

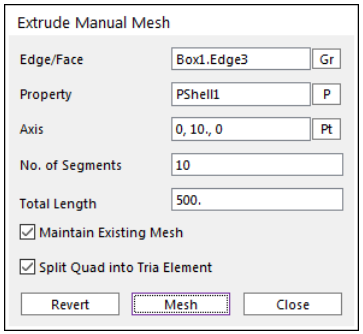


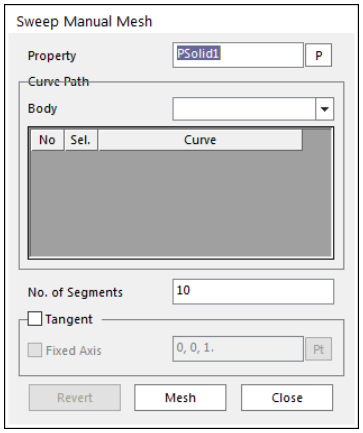
Manual Mesh

❖ Manual Mesh

: Especially, Extrude & Sweep function will be updated







A. Extrude

: When user run an extrude manual mesh on the edge(or surface) geometry, mesher will generate the shell(or solid) mesh result along the defined direction vector and attached that mesh result to the original solid mesh data.

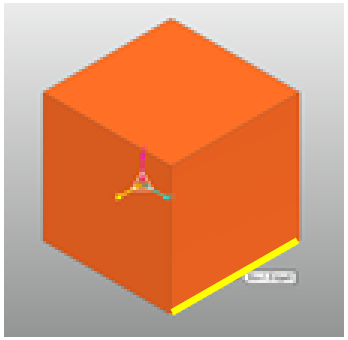
B. Sweep

: When user run a sweep manual mesh, user can create the sweep solid mesh using shell mesh along the defined geometry curve data. Especially, using helix geometry curve, user can create the FE solid mesh data as like a coil spring shape.

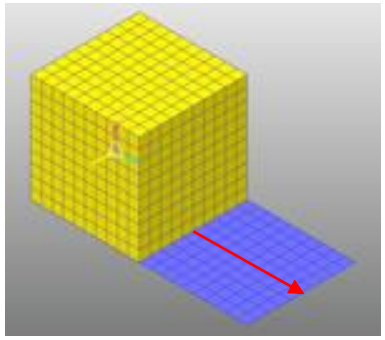
Manual Mesh

- How to use the new options
  - **Manual Mesh – Extrude**
    - ✓ User can create 2D mesh using an edge geometry.
    - ✓ User can create 3D mesh using an surface geometry.
    - ✓ If 'Maintain Existing Mesh' is checked, then the extruded mesh is merged to the original mesh.

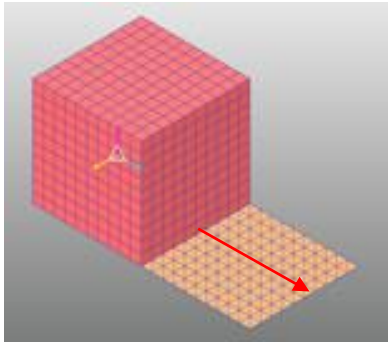
• **Pick the Edge**



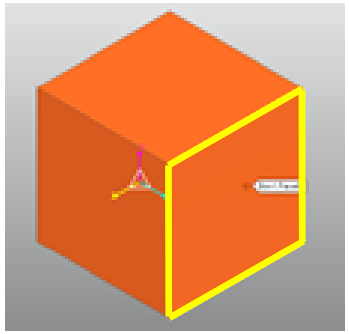
▶ ***Solid8 + Shell4***



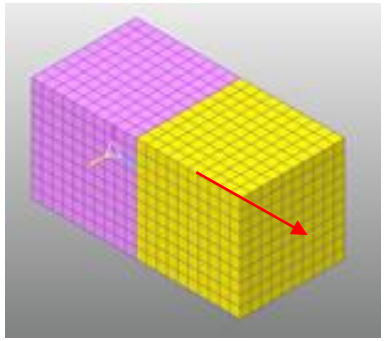
▶ ***Solid8 + Shell3***



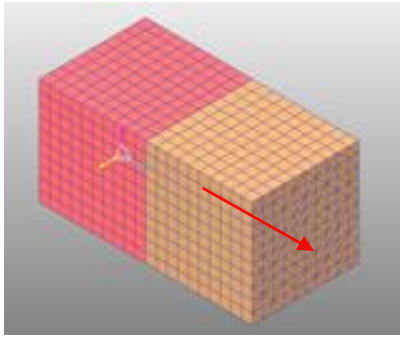
• **Pick the Surface**



▶ ***Solid8 + Shell8***

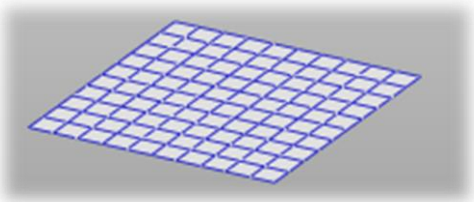


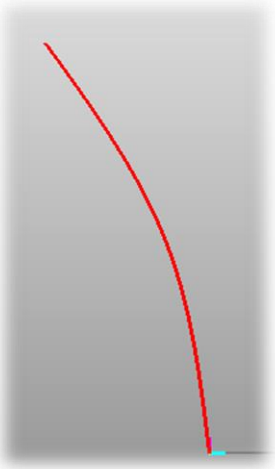
▶ ***Solid8 + Shell6***



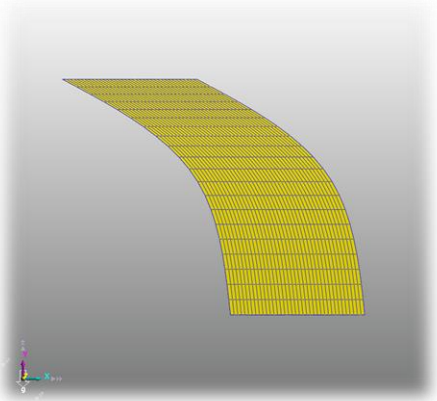
Manual Mesh

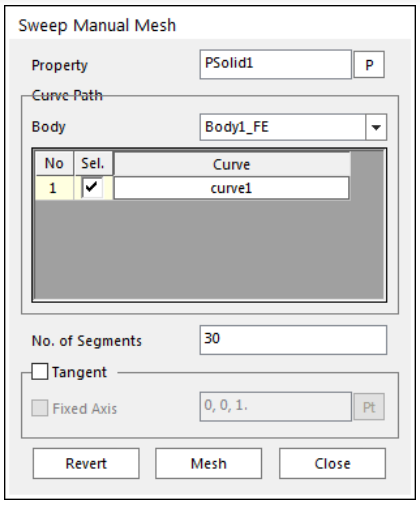
- How to use the new options
  - **Manual Mesh – Sweep(1)**
    - ✓ mesher can create the 3D mesh model using 2D mesh and a curve.
    - ✓ The end point of the curve should be on the surface of 2D mesh.



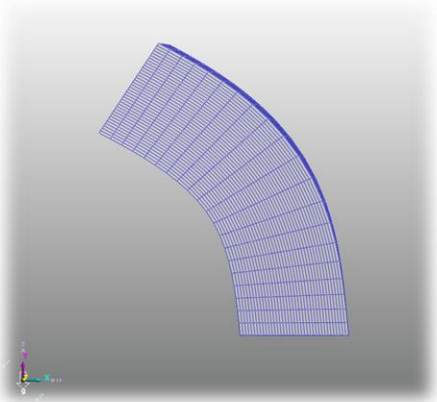


Uncheck Tangent



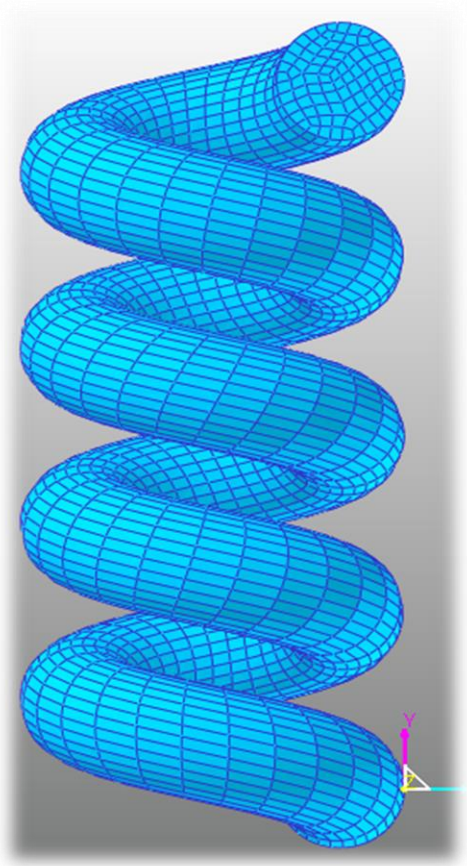


Check Tangent

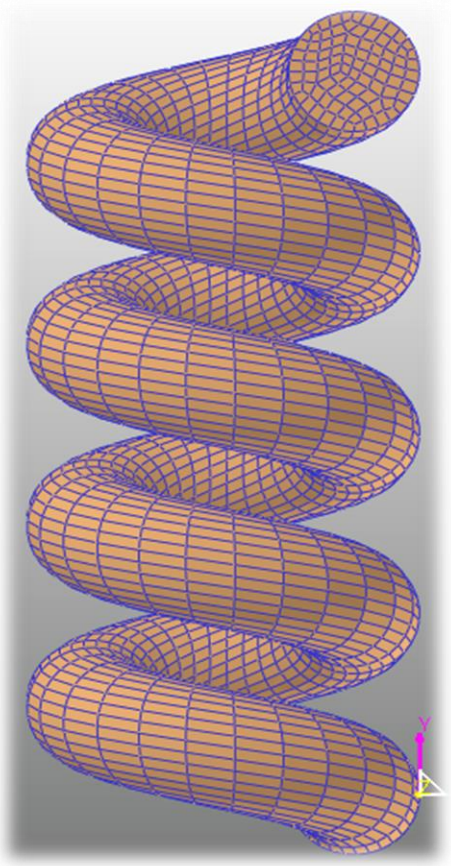


Manual Mesh

- How to use the new options
  - **Manual Mesh – Sweep(2)**
    - ✓ User can create a coil spring shaped solid mesh using a shell mesh and helix curve.

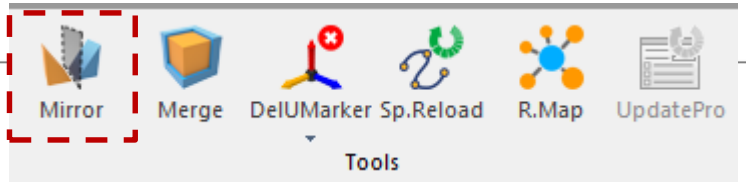


Fixed Axis: Uncheck



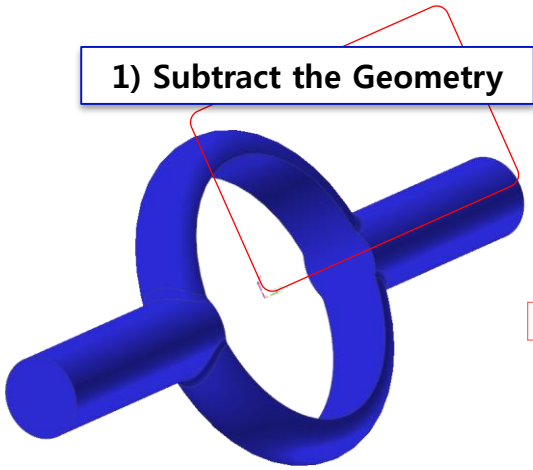
Fixed Axis: Check (0,1,0)

Mesh Functions

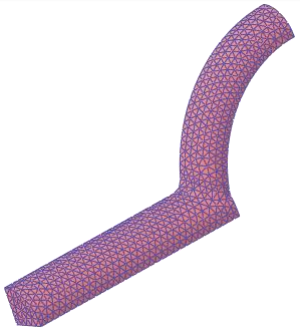


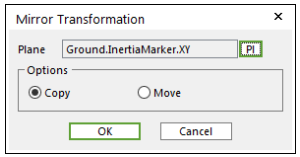
1. New Mesh Function(1)
   1) Mirror Function
      - ✓ Using [Home] > [Tools] > [Mirror] function in V9R1, user can create the symmetric FFlex body.
      - ✓ Please see the below example

1) Subtract the Geometry

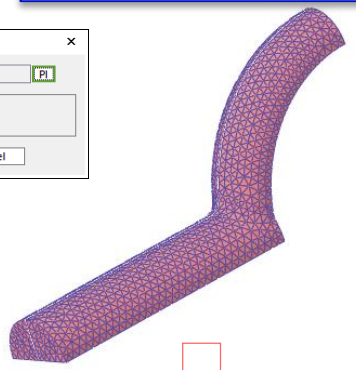


2) Mesh 1/8

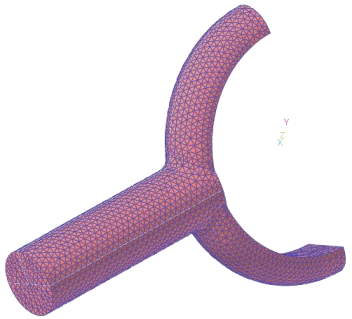




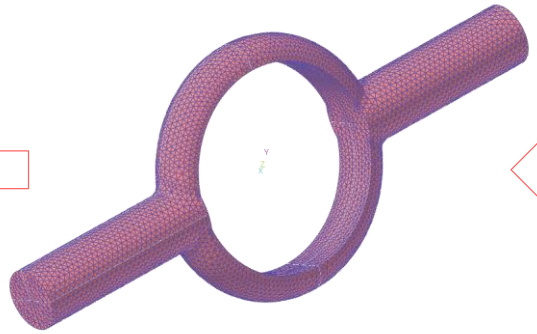
3) Make ¼ by Mirror



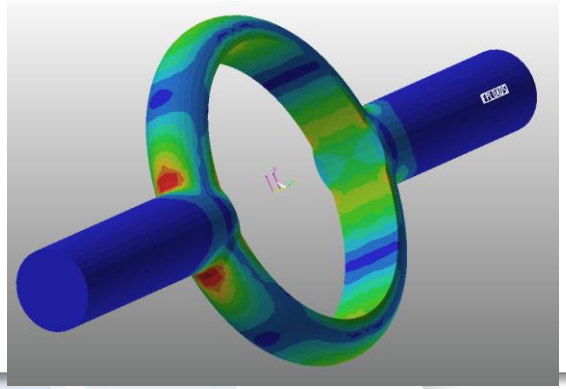
4) Make ½ by Mirror



5) Make whole by Mirror



6) Merge FFlex bodies

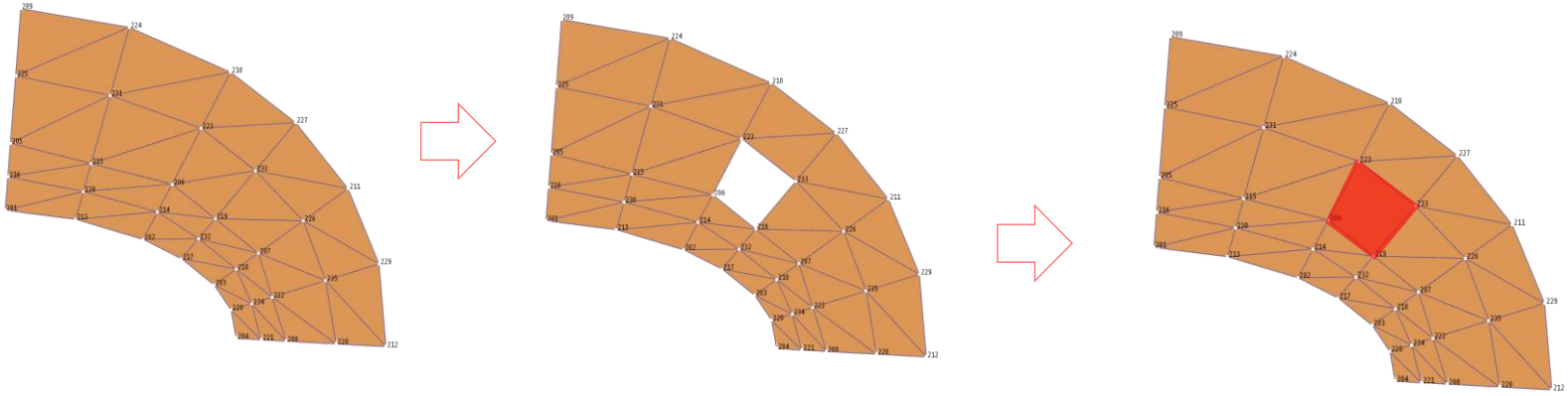


Mesh Functions (Element Create/Delete)

1. New Mesh Function(2)

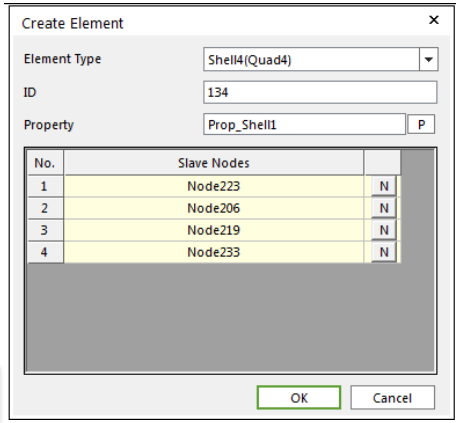
1) Element Create/Delete

- ✓ User can remove the specific element in a FFlex body, then create the FE elements manually.
- ✓ For example, user can remove the some elements in the existing FFlex body as below image.
- ✓ Then, user can create the FE elements manually using the existing nodes.



1. Pick the elements to remove
2. Delete the elements (Delete key)
3. Then the selected elements are deleted.

1. Click [Mesher]-[FFlex Edit]-[Element]
2. Select the slave nodes, element type and property to create an element.



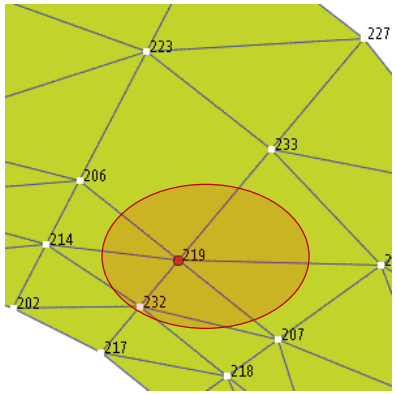
Make an Element manually

Mesh Function (Node modification)

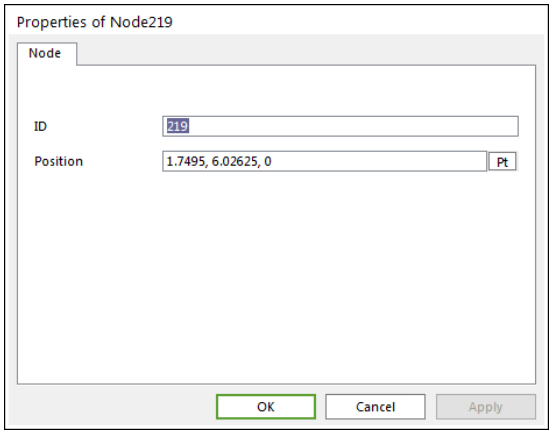
1. New Mesh Function(3)

1) Node Modification

- ✓ : In V8R5, when user modify the node position manually, only node position is updated.
- ✓ : in V9R1, when user modify the node position manually, the connected elements as well as node position are updated automatically.

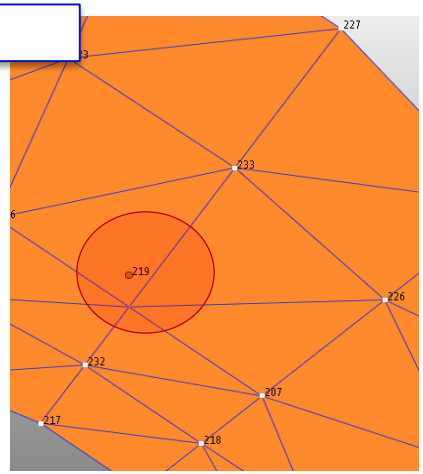


1. Select a node and open the property dialog of node.



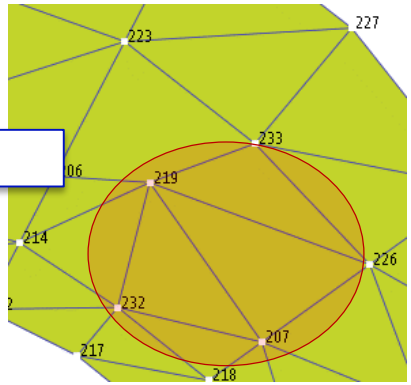
2. Change the node position

V8R5



3. Only node position is updated, so the element shape does NOT change.

V9R1



3. The connected elements are updated when the node position is changed.

V9R1 New Mesher Tutorial