Release 2023 R1 Highlights
Ansys Meshing

New Features (Structures)

- **Explicit Physics Preference (Drop Test Meshing)***
  - Tet Meshing
    - Support for quadratic tet targeting Aspect Ratio and Characteristic Length
    - Characteristic Length actively targeted in PC Tet mesh
    - Thin face/fillet defeaturing
    - Auto-mapping with tri elements of cylindrical faces
- **General Tet Meshing***
  - Feature Suppress (e.g., logos) for Patch Conforming Tetrahedrons Method
- **Welds and Shell Meshing***
  - Robustness improvements over wide range of cases
  - Usability improvements
  - 2nd order shell mesh with linear mid-nodes support
  - Improved quality, small element avoidance
  - Improved error handling for special meshing controls
- **Usability and Automation**
  - Quality Worksheet
  - Detailed Mesh Statistics
  - Settings Improvements
  - Meshing scripting API extended
- **Hex Meshing**
  - Multizone Enhancements
    - Improved warning/error messages
    - Improved robustness and performance
    - Auto-meshing of axisymmetric bodies and tori
  - Feature Suppress (e.g. logos) for Hex Dominant Method
  - SpaceClaim Meshing Enhancements
- **Feature Detection**
  - Progress Bar with Interrupt Function added
  - Performance enhancements
- **Mechanical Beta**
  - Weld Meshing with solid body connections
  - Stacker 2.5D Meshing enhanced for performance, robustness and quality
  - Logo Detection performance improved
  - Sweep “Automatic Thin” supports # divisions for multibody parts

* Major Focus

©2022 ANSYS, Inc. / Confidential

New Features (Fluids, Others)

- **Fluent Meshing**
  - Watertight Meshing
    - Multizone meshing*
    - Fluent/CFX selection in “Generate Volume Mesh”
    - Usability enhancements
  - Fault Tolerant Meshing
    - CFD surface mesh / Conformal mesh
    - Fill without size field
  - Rapid Octree
  - Mesh diagnostic tools
  - Solver Poly Remesh (Beta)*
- **TurboGrid**
  - Extend Blade CAD features
  - Add Blade Blends
  - Adjust Stagger Angles
  - Support for NDF file import
  - New Mesh Quality Metrics for Hybrid Meshes
- **SpaceClaim Meshing**
  - Usability improvements
  - Performance/Robustness/Quality improvements
- **PyPrimeMesh**
  - Surface meshing (Threaded parallel)
  - Wrapper – Fault tolerant meshing (Threaded parellel)
  - Connections
  - Volume meshing (Distributed parallel)
  - Graphics (PyVista)

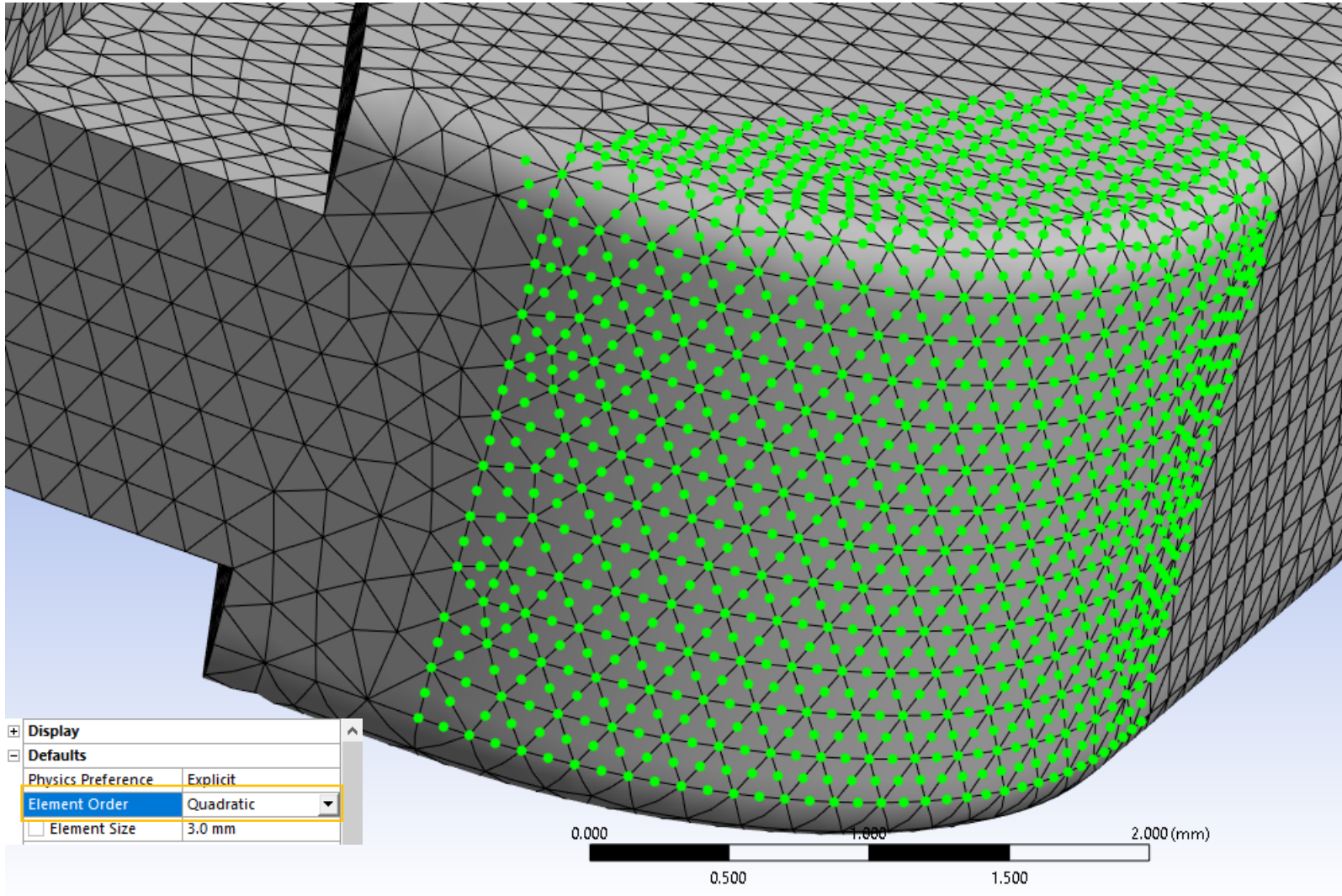
* Major Focus

©2022 ANSYS, Inc. / Confidential

Explicit Physics Preference (Drop Test Meshing)

Patch Conforming Tetrahedrons : Quadratic Tet for Explicit

- Explicit Physics Preference (targeting Aspect Ratio quality improvements) was supporting only linear mid-side-nodes in 22R2
- Extended to full quadratic elements in 23R1
- Important for problems with thin structures where only single elements can be used to resolve thickness
  - E.g. Smartphone drop-test



©2022 ANSYS, Inc. / Confidential

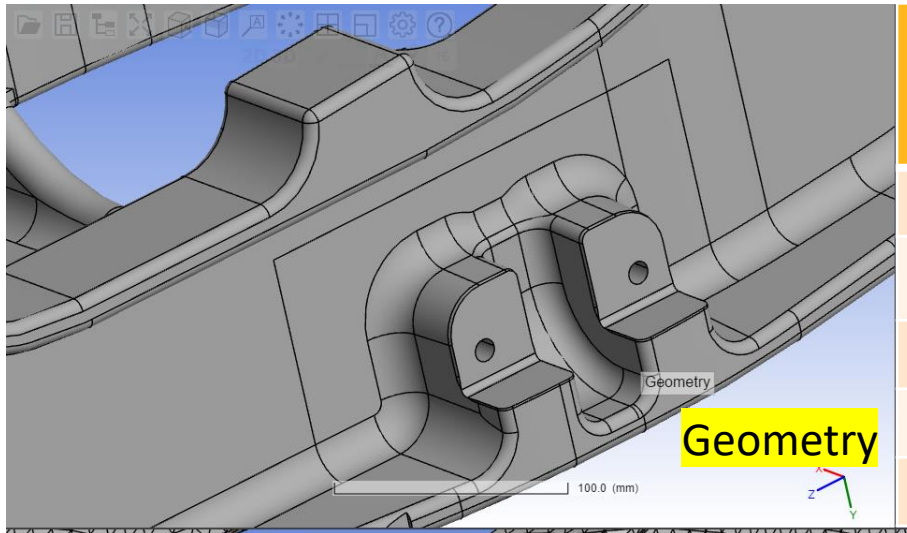
Explicit Targeting of Char. Length and Aspect Ratio Improved

Aspect Ratio ⇩

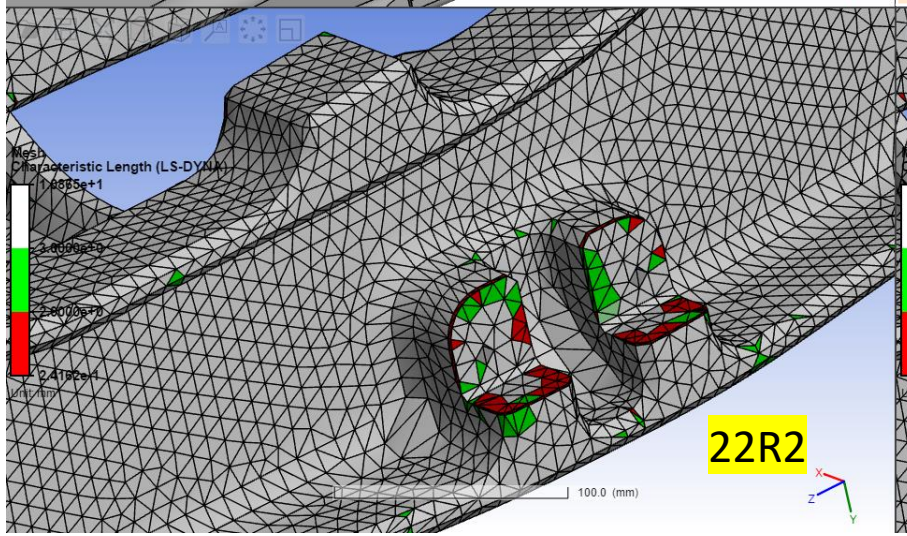
Char. Length ⇧

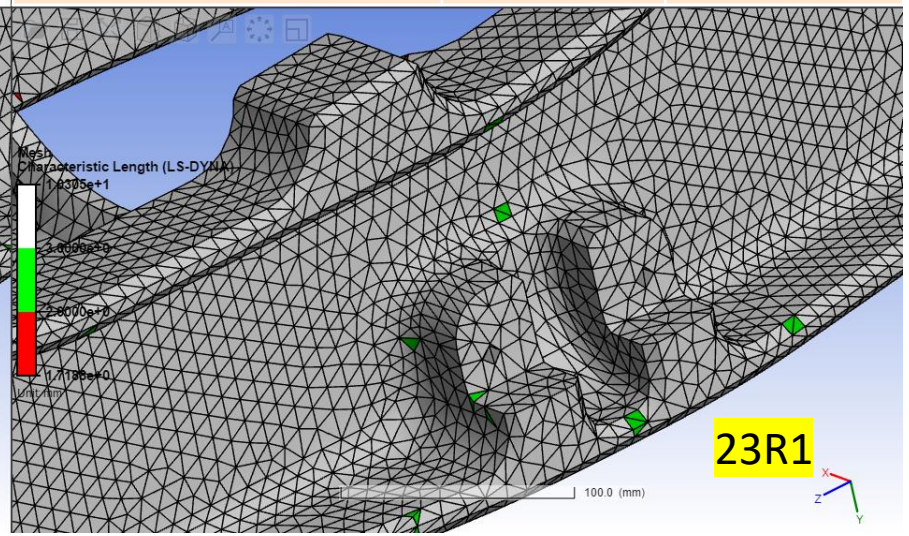
Features ⇔

Improved small element removal and fillet meshing with mapped mesh controls



	22R2	23R1
Min CL	0.242mm	1.72mm
# Elements < Target CL	669	11
Max AR	18.1	5.3
# Elements > AR = 5	237	2
# Elements > AR = 10	41	0





©2022 ANSYS, Inc. / Confidential

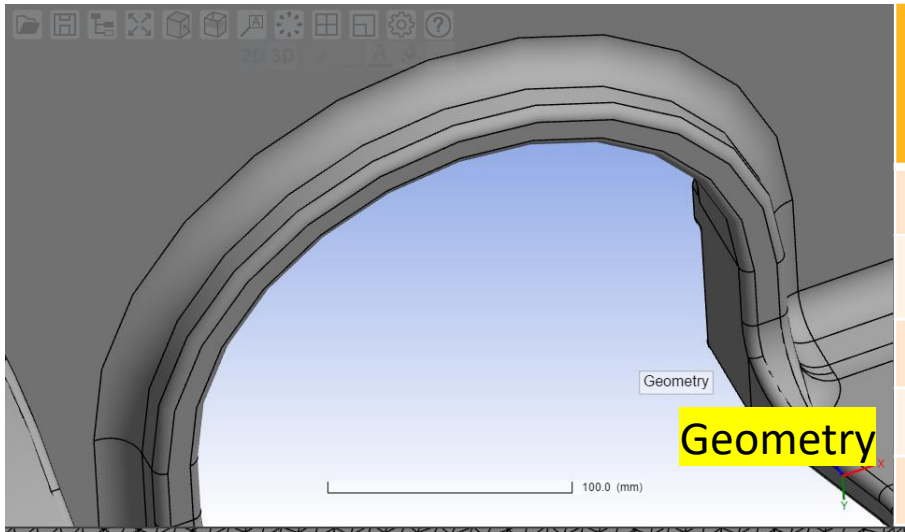
Explicit Targeting of Char. Length and Aspect Ratio Improved

Aspect Ratio ⇓

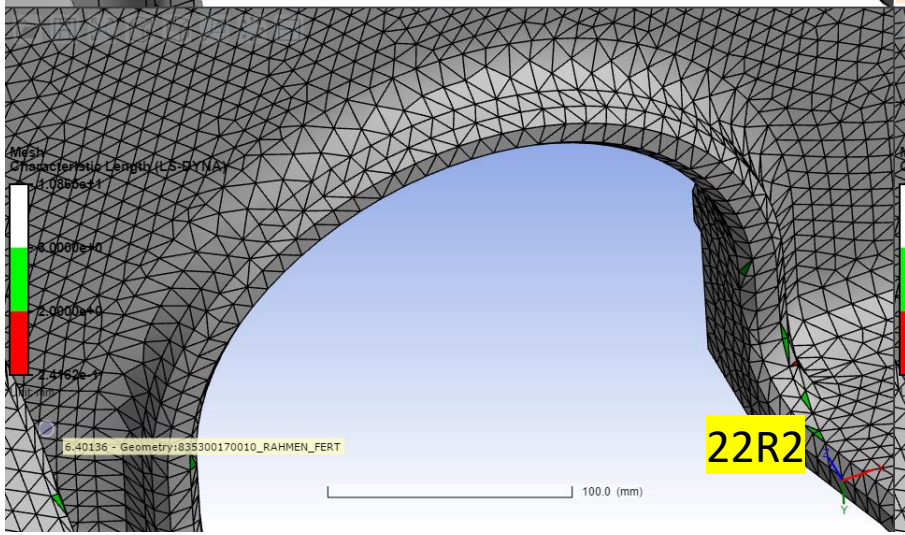
Char. Length ⇑

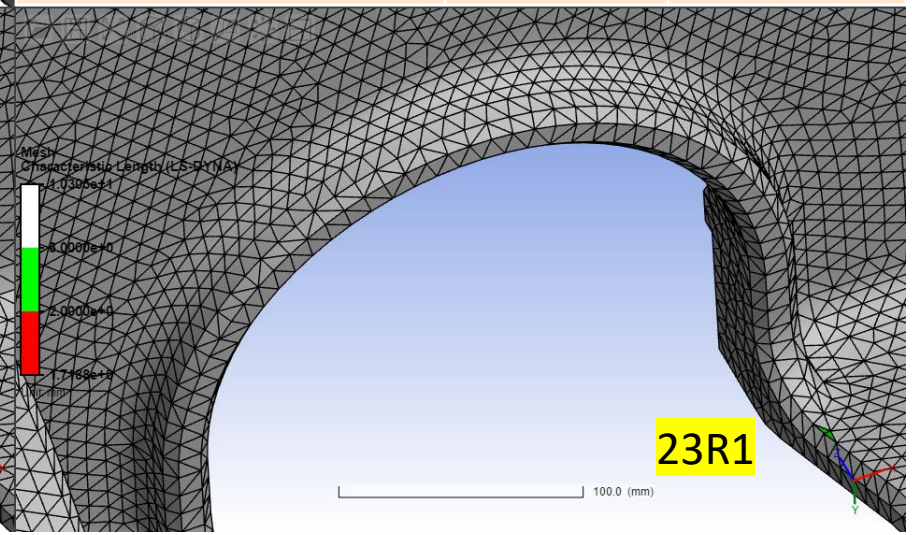
Features ⇔

Improved small element removal and fillet meshing with mapped mesh controls



	22R2	23R1
Min CL	0.242mm	1.72mm
# Elements < Target CL	669	11
Max AR	18.1	5.3
# Elements > AR = 5	237	2
# Elements > AR = 10	41	0





©2022 ANSYS, Inc. / Confidential

Patch Conforming Tetrahedrons : Improved Quality/Fidelity

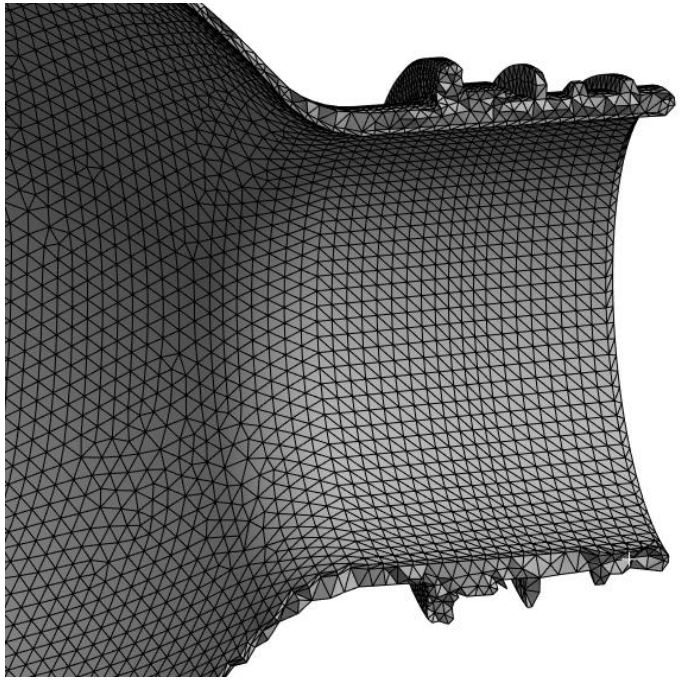
- Improvements to Patch Conforming Tetrahedrons under Explicit Physics Preference*
- Targeting Max Aspect Ratio AND Min Characteristic Length AND retain features/fillets AND **mapped mesh**
  - Surface mesh clean-up automatically to target CL and AR
  - Auto-defeaturing of very thin fillets/faces
- Map Meshing improvements
  - Auto-mapping of cylinders (no control or detection required)
  - Improvements to allow some defeaturing with mapped mesh controls

Aspect Ratio ↓

Char. Length ↑

Features ↔

Auto-Mapped Cylinders



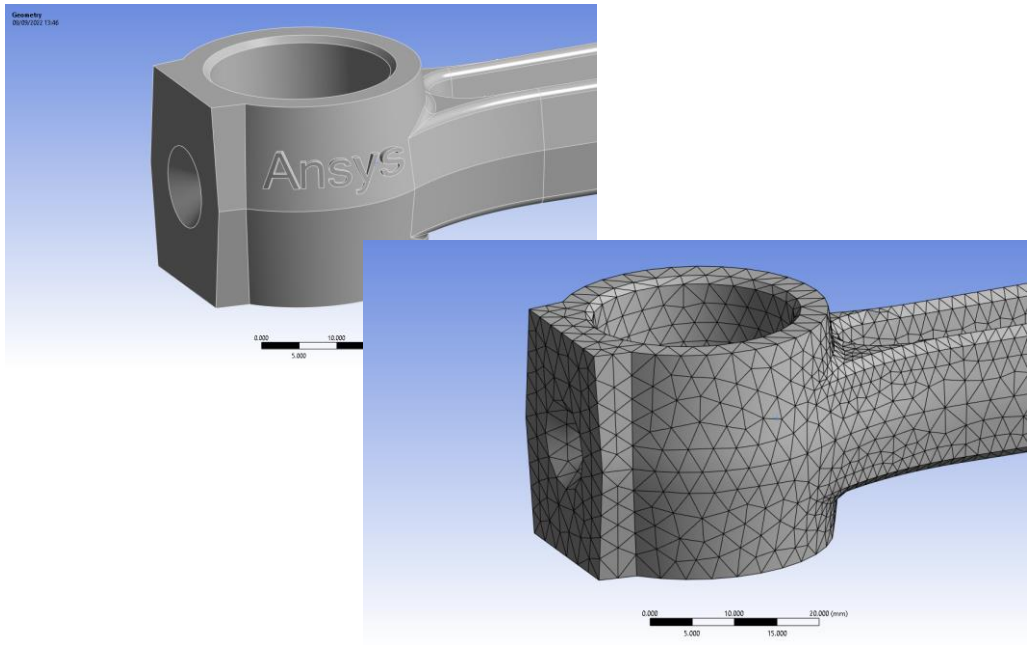
*Explicit Physics Preference enables (a) Targeting for specific qualities with surface mesh clean-up and (b) Sizing to target uniform mesh. Elements produced are still compatible with implicit solver. It can be used for e.g. implicit calculations but user would need to manually set appropriate size controls and likely reduce the targeting criteria to avoid defeaturing heavily.

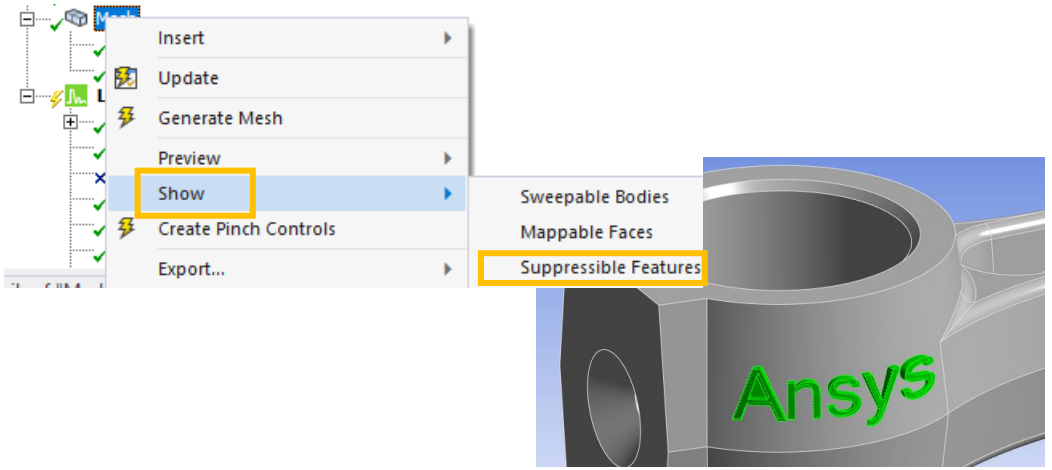
©2022 ANSYS, Inc. / Confidential

General Tet Meshing

Feature Suppress

- Removes embossed and indented features which can cause quality and robustness issues (e.g. logos)
- 3 Scoping Methods
  - Automatic (Features detected and removed based on size)
  - Defeature Faces (Select what to remove)
  - Parent Faces (Select faces that enclose the features to remove)
- After scoping user can preview which features will be suppressed during meshing via Show → Suppressible Features
  - After meshing, user can also use Named Selection Diagnostics to show Defeatured Faces
- Protected Topology Named Selections will be protected
- Faces/Edges with local sizing scoped will automatically be protected

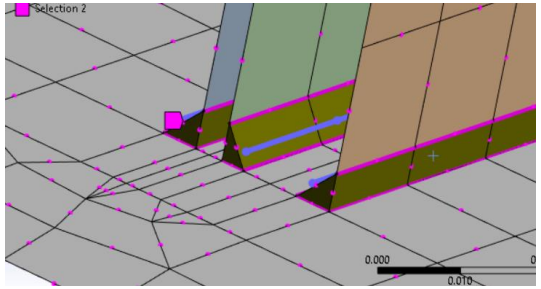


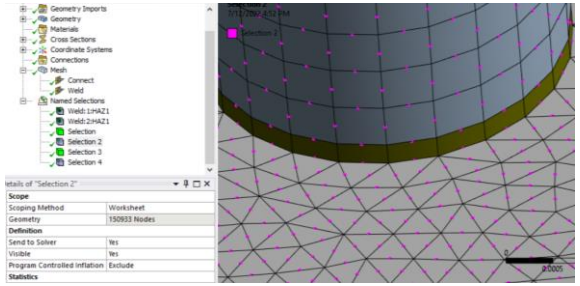


©2022 ANSYS, Inc. / Confidential

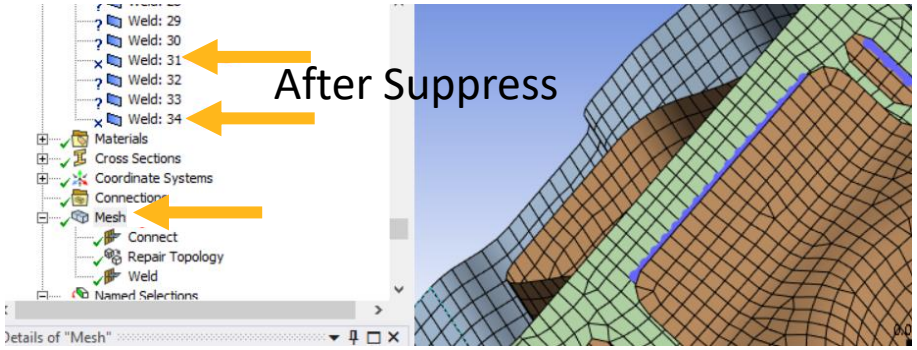
Welds and Shell Meshing

Weld Meshing Enhancements (Batch Connections)

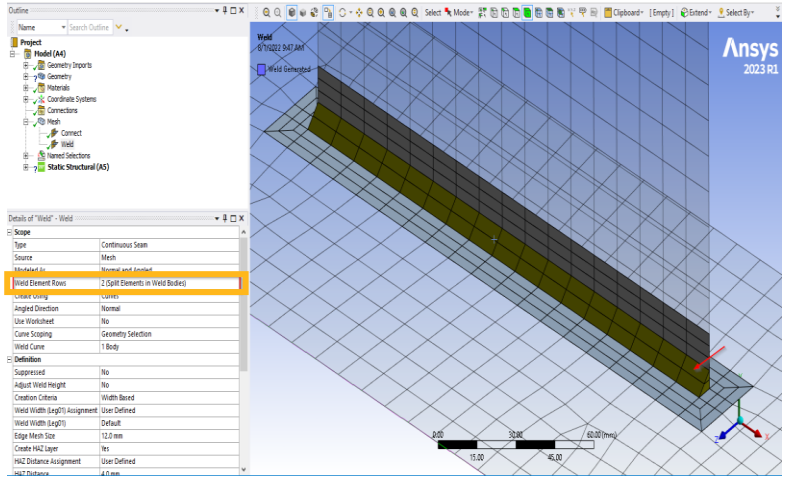




Support for 2nd-order mesh with straight-sided mid nodes



Suppress or un-suppress weld geometry without obsoleting mesh status

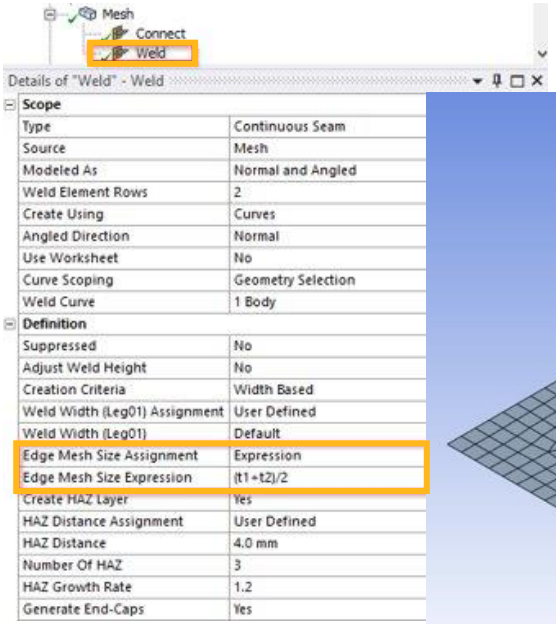


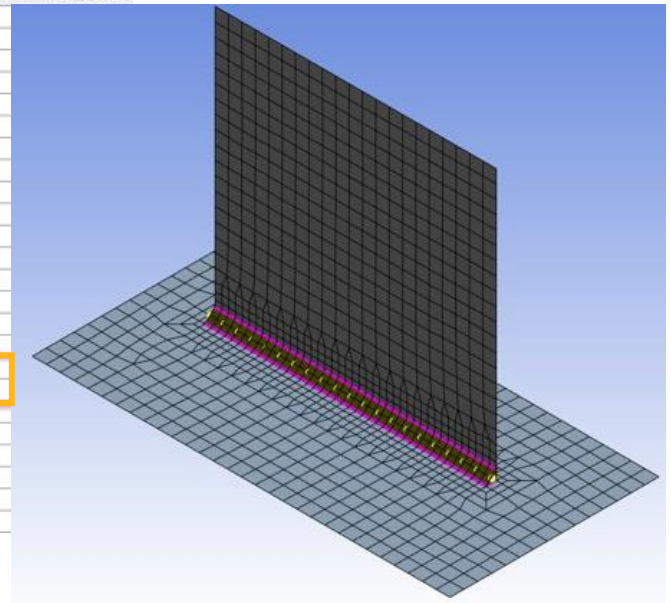
New UI option added "Weld Element Rows" to split elements from weld faces.

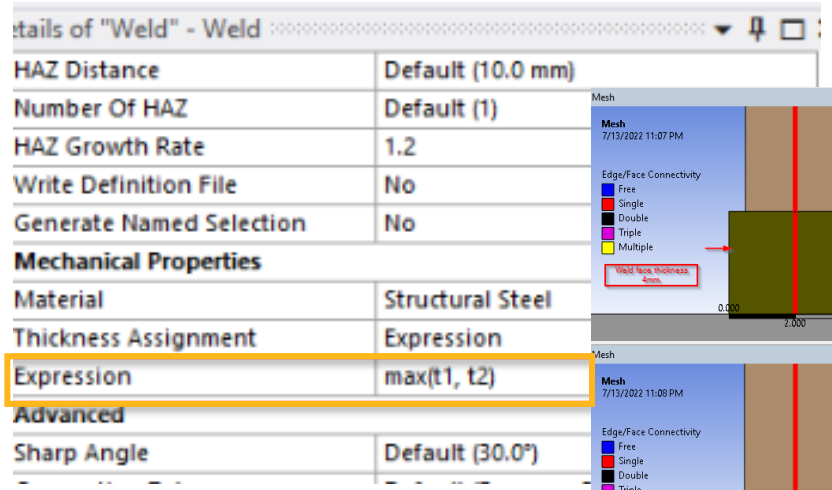
©2022 ANSYS, Inc. / Confidential

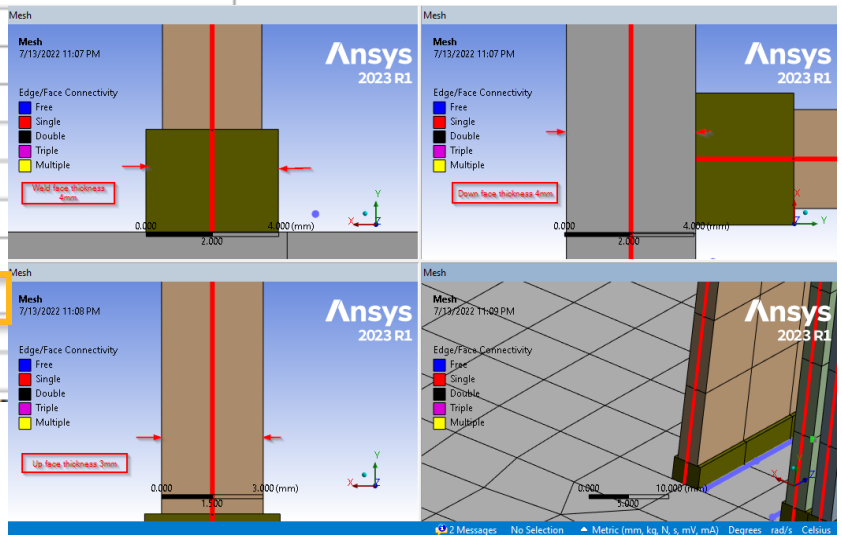
Weld Meshing Enhancements (Batch Connections)
Improvements to Weld Expressions

Expressions support added for:
min(t1,t2), max (t1,t2), scientific numbering (1e-5),
Weld edge mesh size







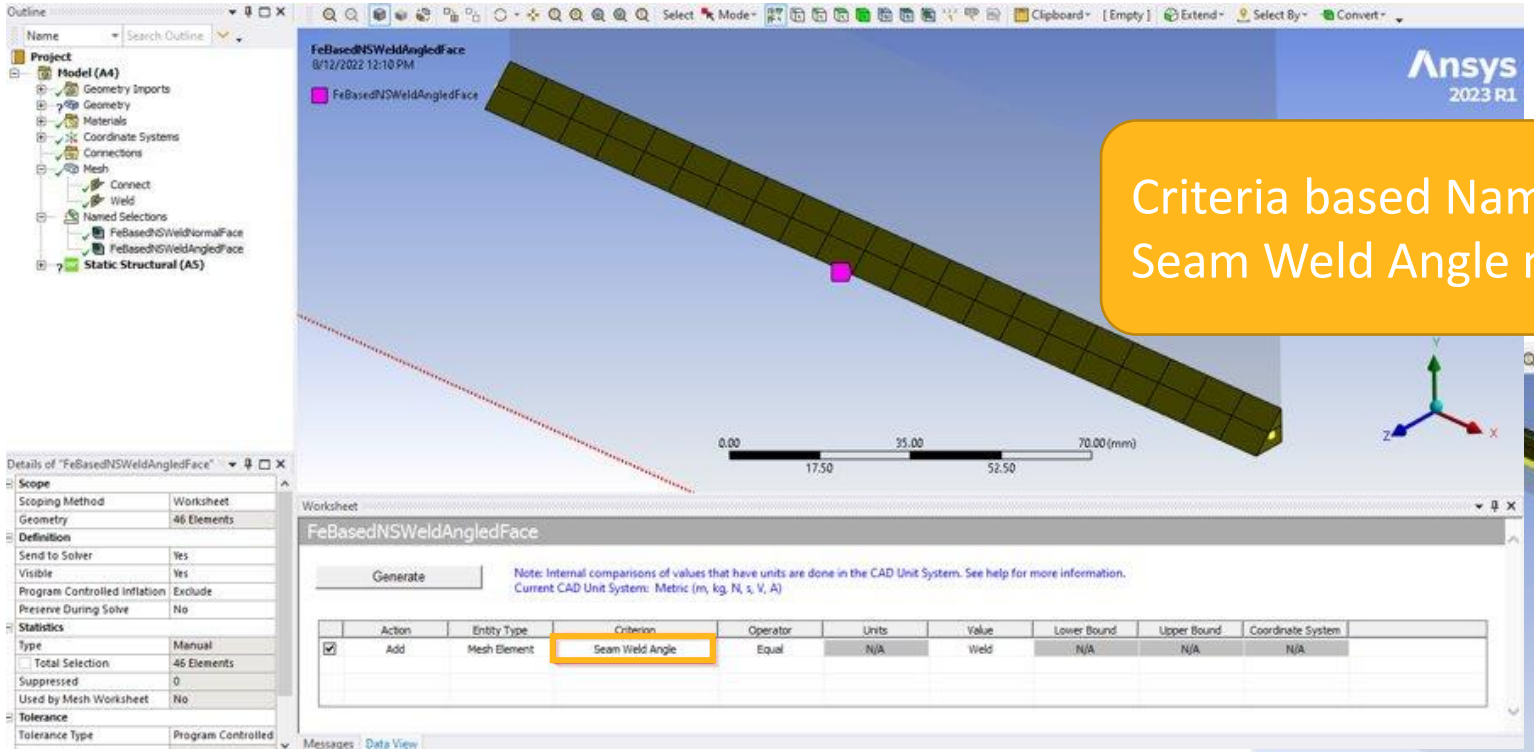


Edge Mesh Size defined using User defined Expression

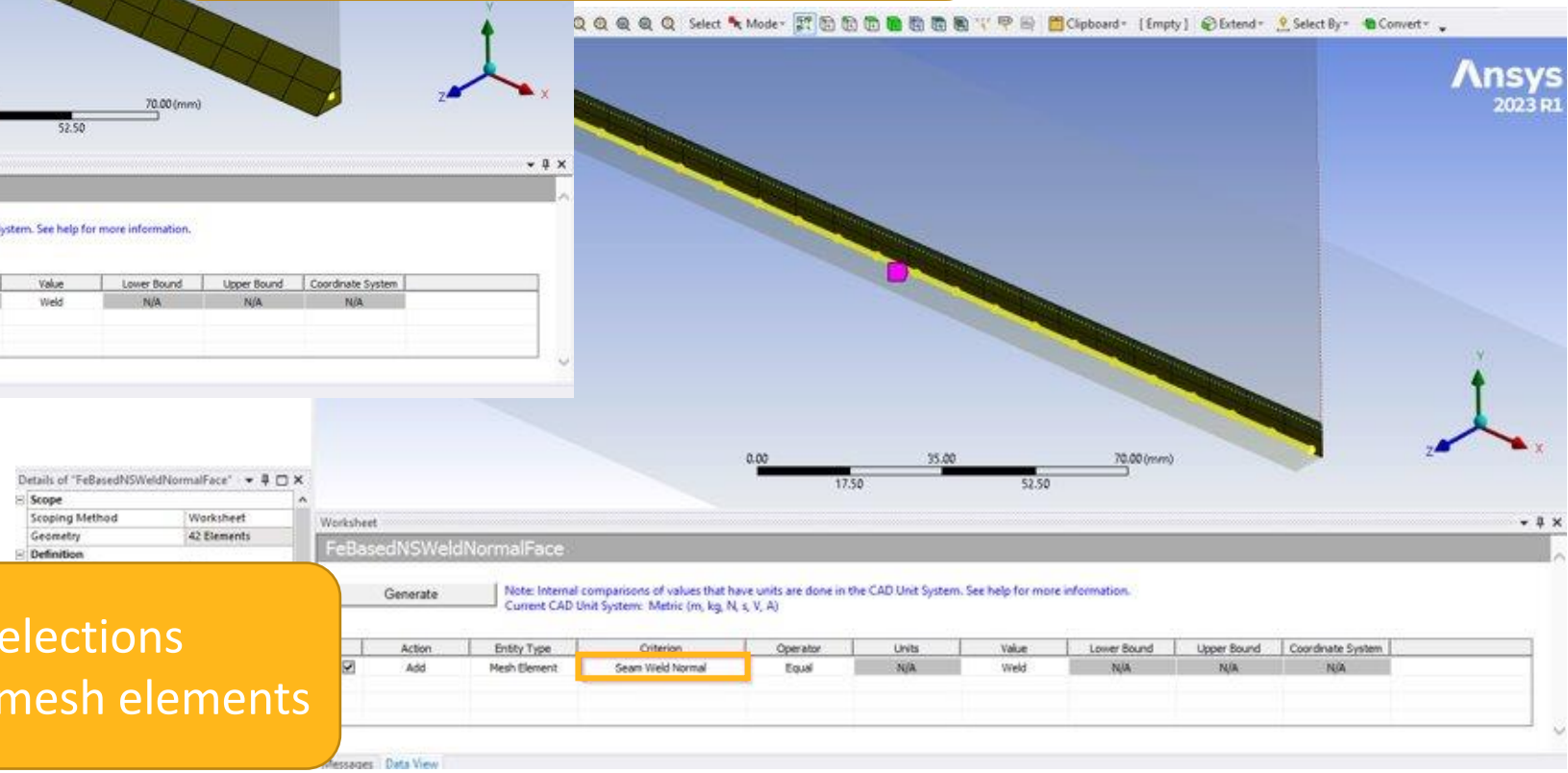
Resultant thickness = 4mm based on down face thickness 4mm

©2022 ANSYS, Inc. / Confidential

Weld Meshing Enhancements (Batch Connections)



Criteria based Named Selections for Seam Weld Angle mesh elements



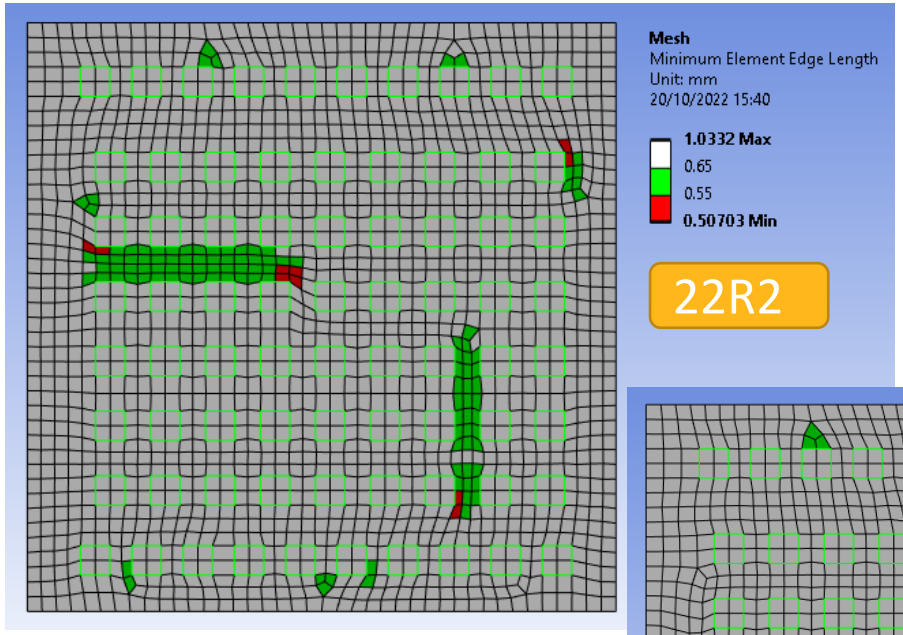
Criteria based Named Selections for Seam Weld Normal mesh elements

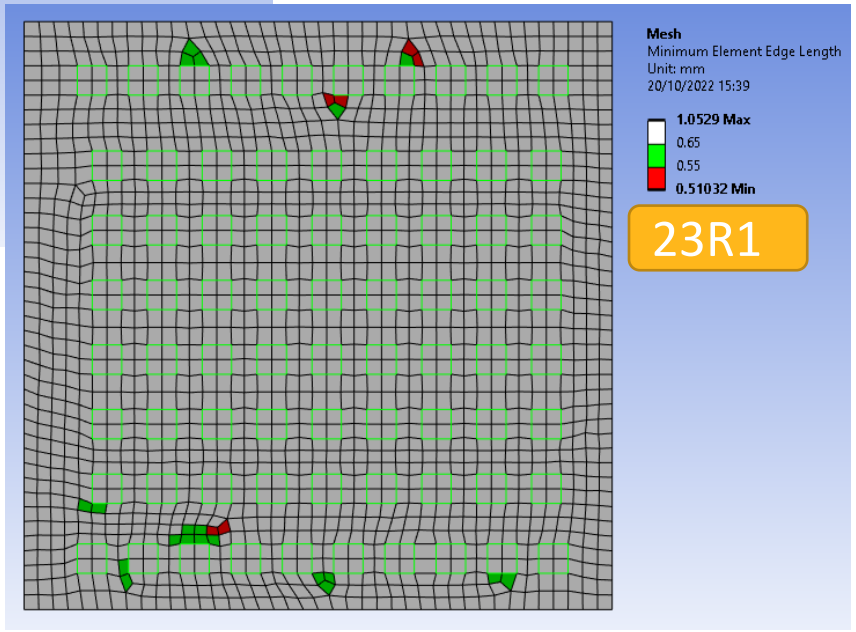
©2022 ANSYS, Inc. / Confidential

Batch Connections Quad Shell Meshing

- Reduction of small elements when Aggressive Tri Reduction is used
- Ongoing improvements focussing on PCB meshing with Stacker (See Beta section)

Batch Connections	
Mesh Based Connection	Yes
Mesh Type	Quadrilaterals
Triangle Reduction	Aggressive
Tolerance For Immediate Connections	Always Use Local
Local Connection Tolerance	Default (0.1 mm)
Local Connection Option	All To All

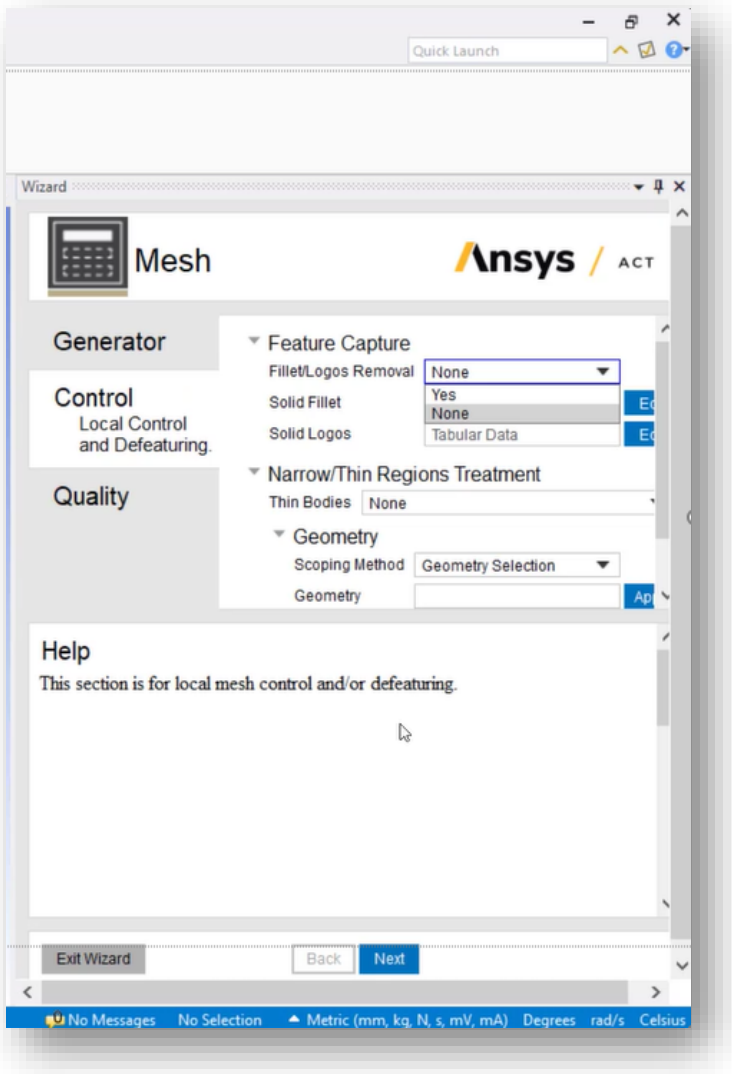




©2022 ANSYS, Inc. / Confidential

Scripting APIs Extended for Meshing

- Support added for scripting advanced settings for local size controls
  - E.g. curvature/proximity etc
- Support for quality worksheet targeting, criteria visibility, …
- Important for building ACT meshing workflows to simplify meshing and automation of best practices

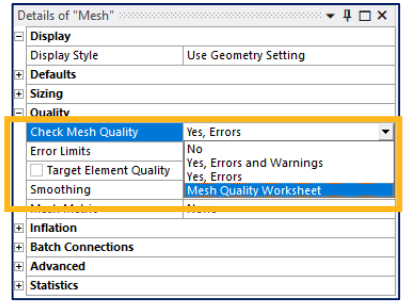


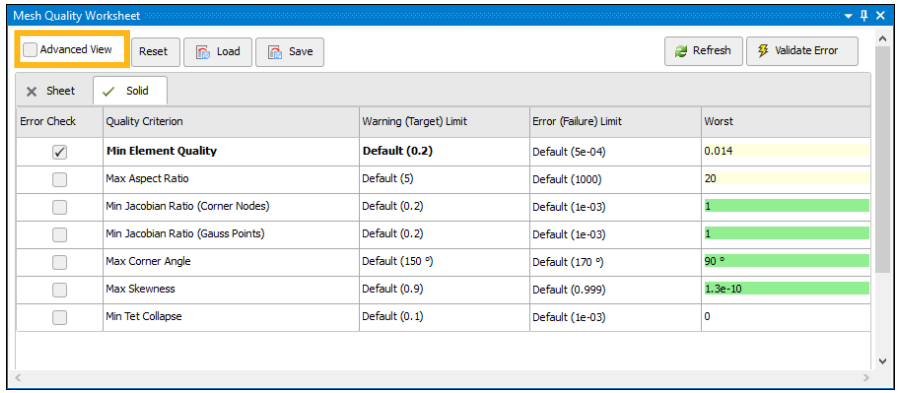
©2022 ANSYS, Inc. / Confidential

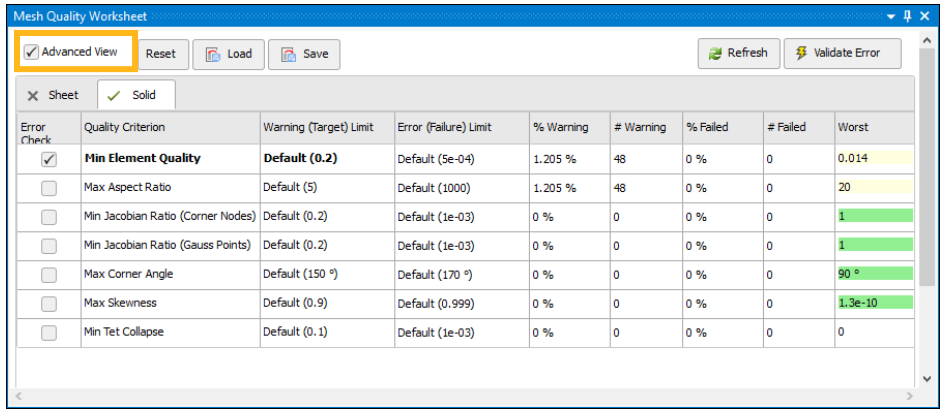
Usability and Automation

Mesh Quality Worksheet

- New option under **Check Mesh Quality**
  - Mesh Quality Worksheet
- When enabled shows a table of Physics Preference-based quality criteria
- Tabs are available for Sheet and Solid bodies
- User can choose which criteria to show/calculate in the User Preferences
- Default and **Advanced View** is available
  - Advanced View shows more detailed information about % and number of elements exceeding warning/error limits







©2022 ANSYS, Inc. / Confidential

Mesh Quality Worksheet Explained

Advanced View
Switches on/off visibility of % and # columns

Reset
Resets all to default values

Load/Save
Read and write criteria to *.csv

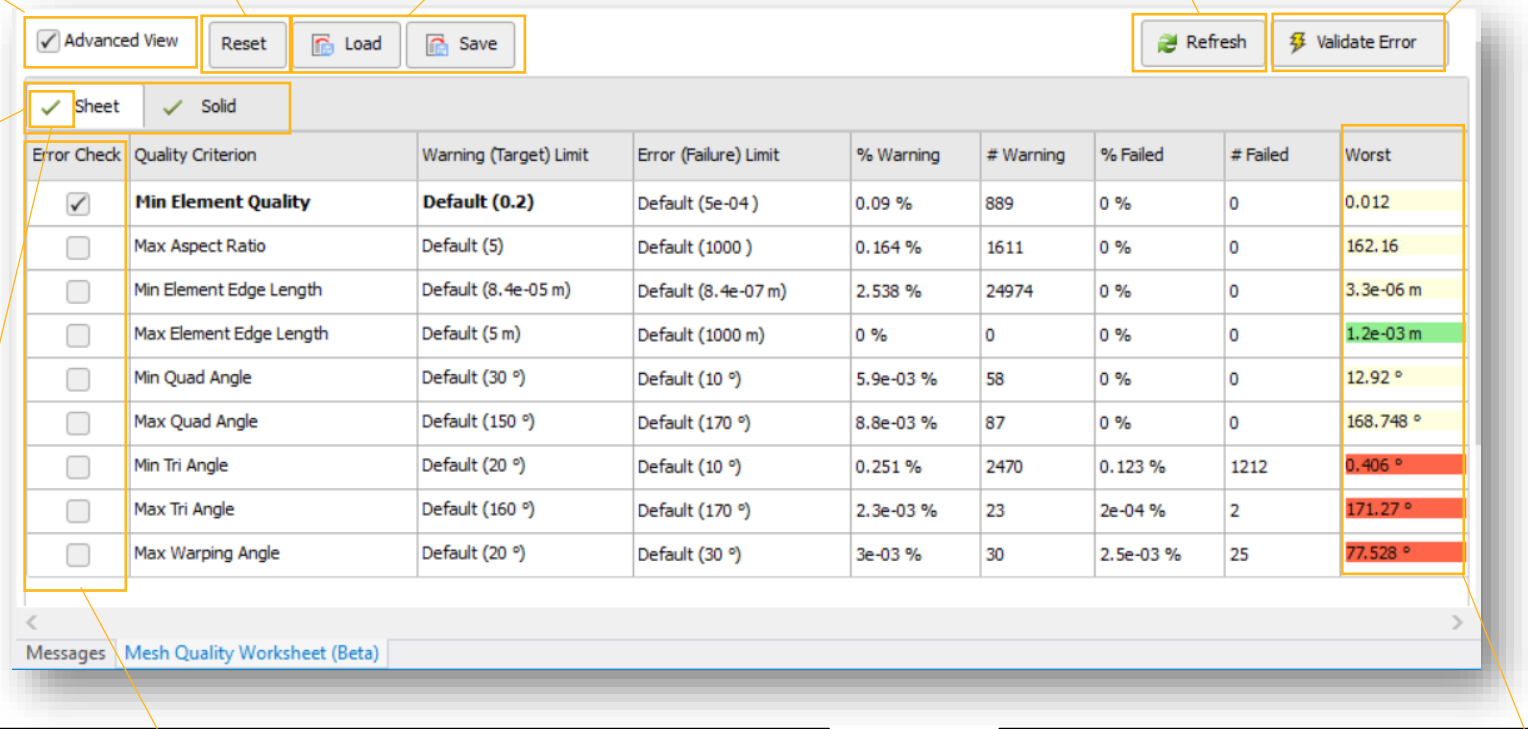
Refresh
Update % and # values based on modified inputs without invalidating mesh

Validate Error
Update % and # values based on modified inputs AND invalidates mesh if "Error Check" is ON for that Criterion

Sheet/Solid
Tabs to show quality for sheet or solid bodies

Status Symbol Tick/Refresh/Cross
Status symbols to indicate if criteria calculations are up to date, need to be refreshed or are not applicable

Bold Quality Criteria
Bold Quality Criteria are those that are sent to the meshing engine and actively participate in meshing



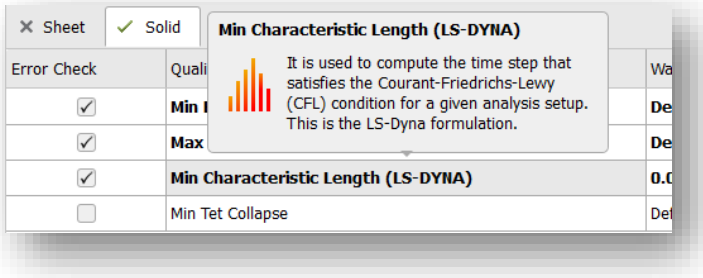
Error Check	Quality Criterion	Warning (Target) Limit	Error (Failure) Limit	% Warning	# Warning	% Failed	# Failed	Worst
☑	**Min Element Quality**	**Default (0.2)**	Default (5e-04)	0.09 %	889	0 %	0	0.012
☐	Max Aspect Ratio	Default (5)	Default (1000)	0.164 %	1611	0 %	0	162.16
☐	Min Element Edge Length	Default (8.4e-05 m)	Default (8.4e-07 m)	2.538 %	24974	0 %	0	3.3e-06 m
☐	Max Element Edge Length	Default (5 m)	Default (1000 m)	0 %	0	0 %	0	1.2e-03 m
☐	Min Quad Angle	Default (30 °)	Default (10 °)	5.9e-03 %	58	0 %	0	12.92 °
☐	Max Quad Angle	Default (150 °)	Default (170 °)	8.8e-03 %	87	0 %	0	168.748 °
☐	Min Tri Angle	Default (20 °)	Default (10 °)	0.251 %	2470	0.123 %	1212	0.406 °
☐	Max Tri Angle	Default (160 °)	Default (170 °)	2.3e-03 %	23	2e-04 %	2	171.27 °
☐	Max Warping Angle	Default (20 °)	Default (30 °)	3e-03 %	30	2.5e-03 %	25	77.528 °

Error Check
During validation after meshing or when "Validate Error" is pressed, the mesh will only be invalidated if failure limits are exceeded AND this check box is enabled. Useful for e.g. DoE studies where strict quality limits are required.

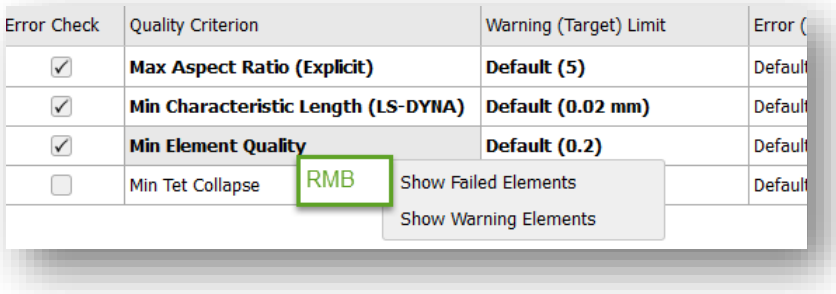
Worst
Shows worst metric for that criteria and highlights by colour if this criteria is in "Good", "Warning" or "Failure" State

©2022 ANSYS, Inc. / Confidential

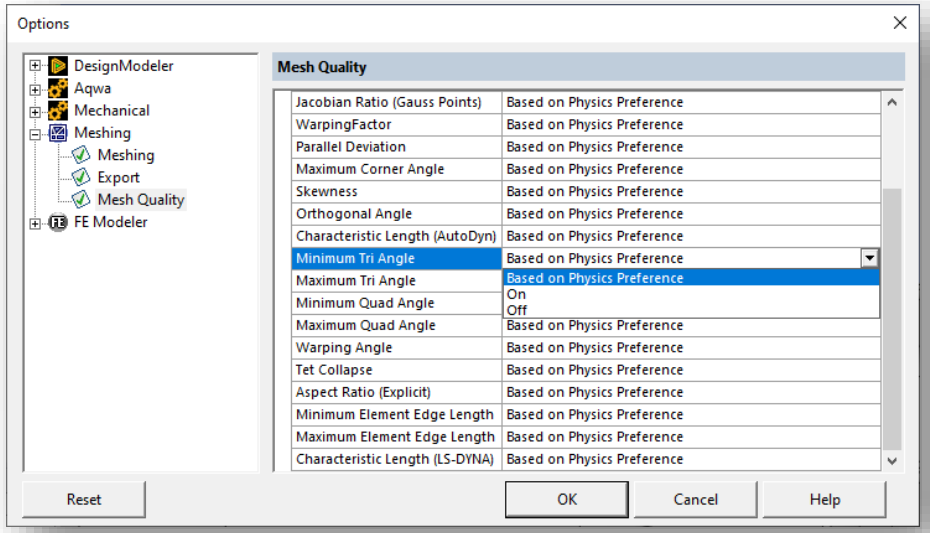
Mesh Quality Worksheet – Additional Features



Tooltips for quick documentation



Right-click on a quality criterion to quickly generate NS element groups for failed/warning elements



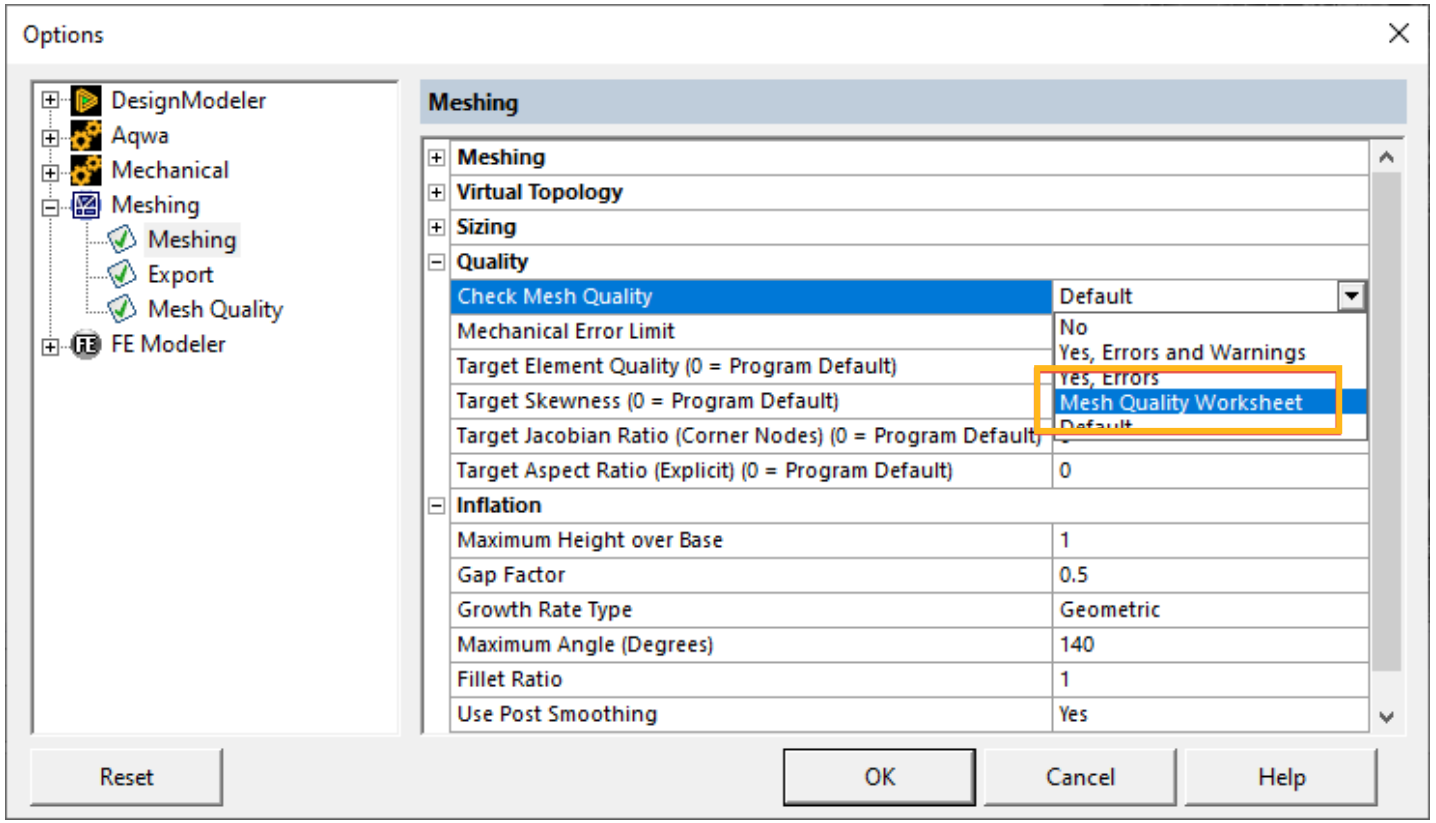
Visibility of quality metric is now configurable in the Quality Worksheet from the Meshing Options

- Program controlled by default per physics preference.

©2022 ANSYS, Inc. / Confidential

Preference to see Mesh Quality Worksheet by Default

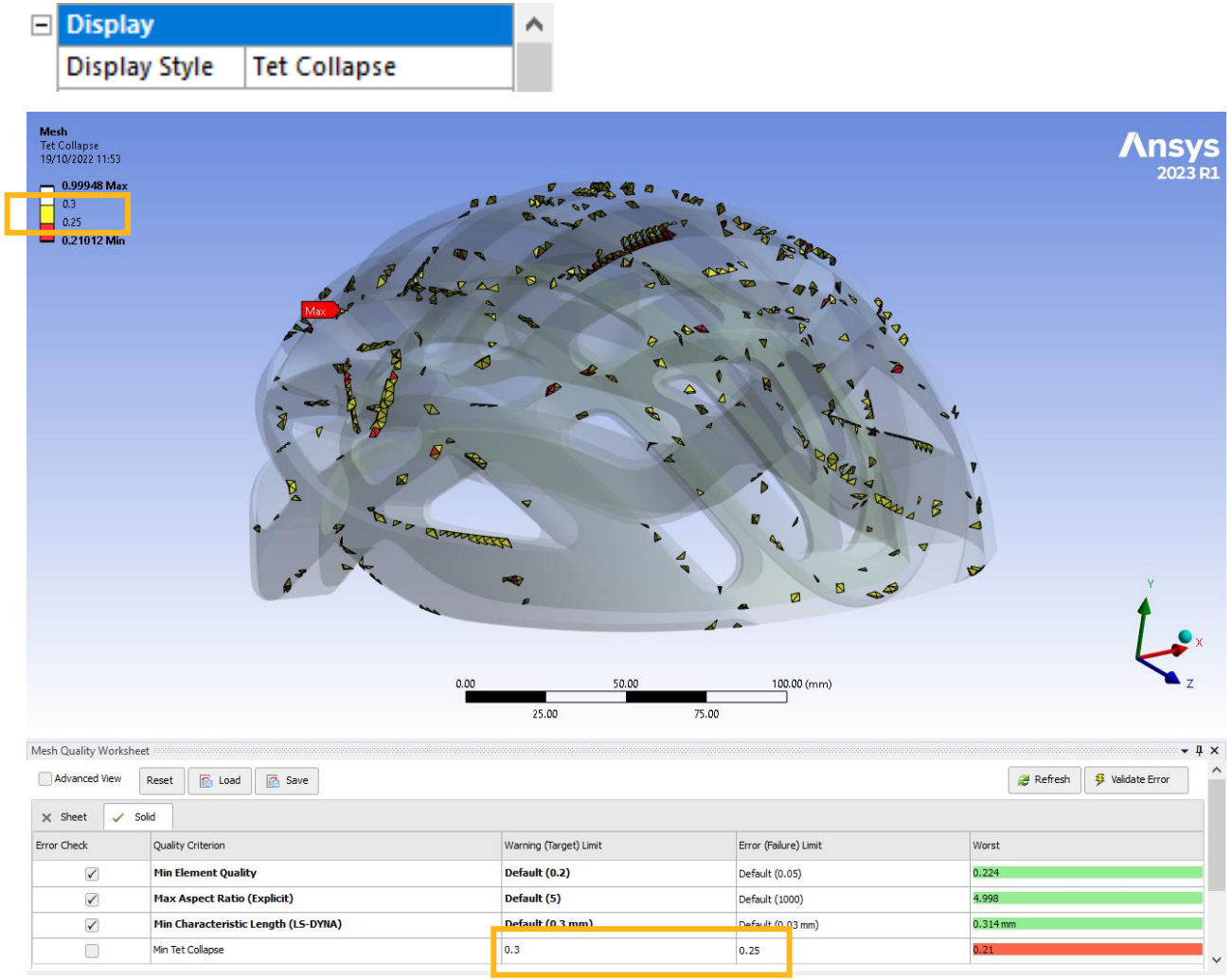
- User Preference added to make Quality Worksheet default choice in new Sessions



©2022 ANSYS, Inc. / Confidential

Contour Banding in Quality Worksheet Mode

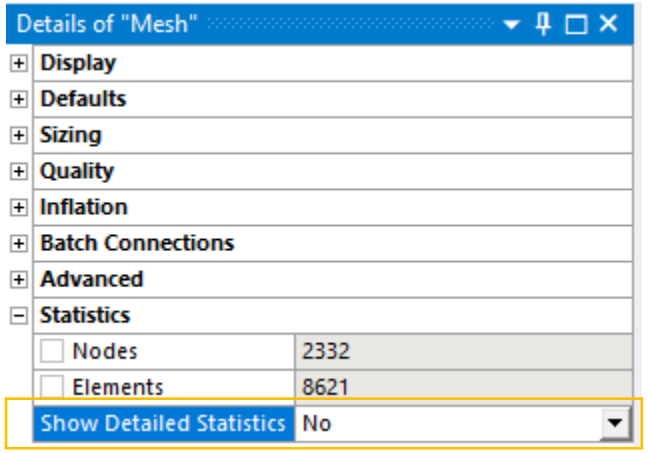
- When Quality Worksheet is Enabled Display Style contours are modified
- Warning and Error limits are used to populate a minimal Legend
- Warning Elements are shown as **yellow**
- Error Elements are shown as **red**

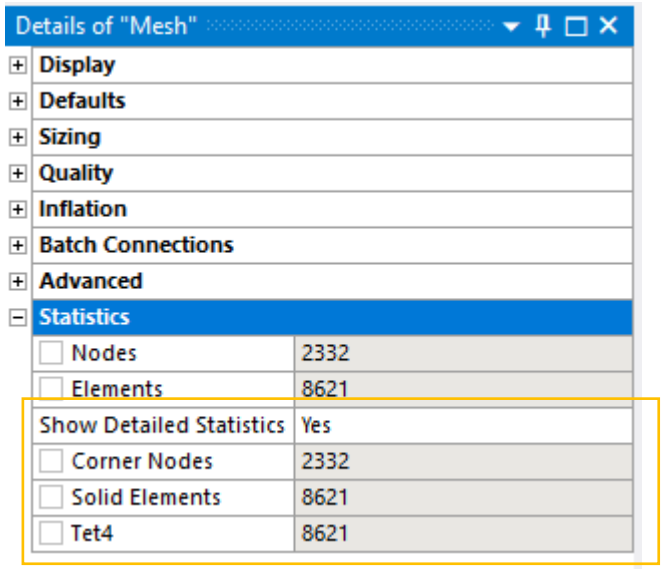


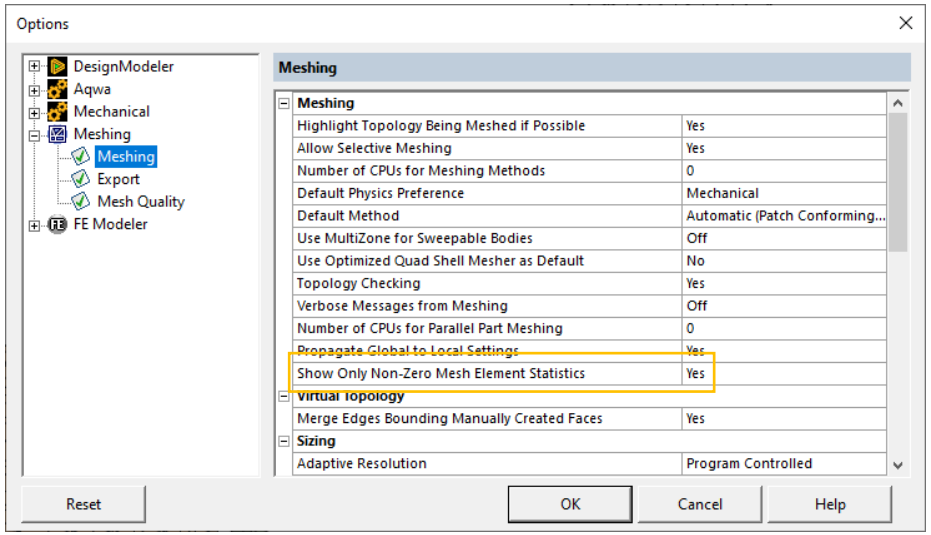
©2022 ANSYS, Inc. / Confidential

Detailed Mesh Statistics

- Prior to 23R1 Statistics only gave Node and Element Count
- **Show Detailed Statistics** now shows data for all element types
  - Default value is No
  - By default, Mechanical only shows elements that exist in the mesh
  - Preference in Mesh Options to show zero values
    - Useful to detect/flag presence of unwanted element types during design studies (e.g. pyramids)
    - Default value is No



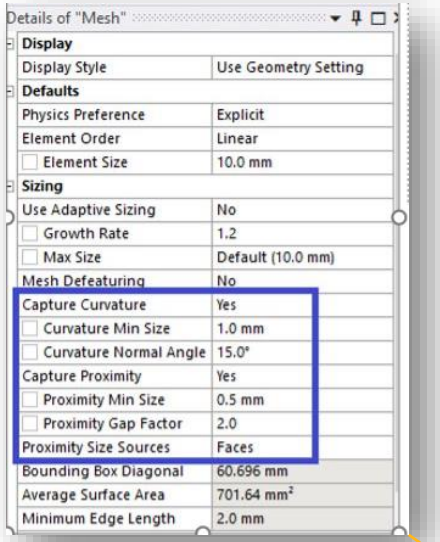




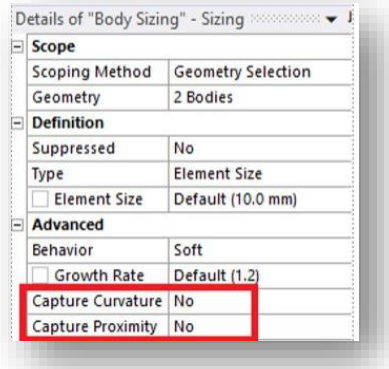
©2022 ANSYS, Inc. / Confidential

Default Local Settings

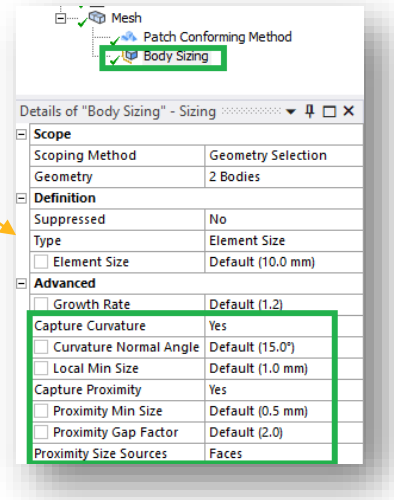
- In 23R1 when user creates a local control (Body, Face, Edge) the initial control will be populated with the Global Settings
  - E.g. if curvature or proximity are enabled globally, they will be enabled by default in any new control created
  - More intuitive outcomes for users



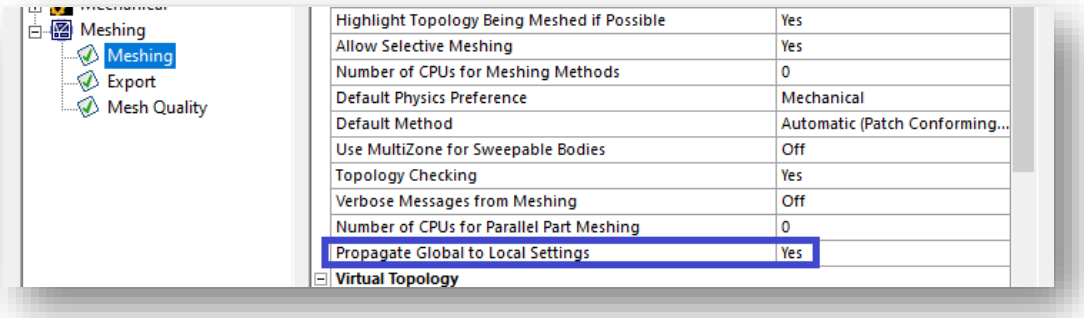
Global Settings



Propagate Global to Local Settings = No



Propagate Global to Local Settings = Yes (Default behavior)

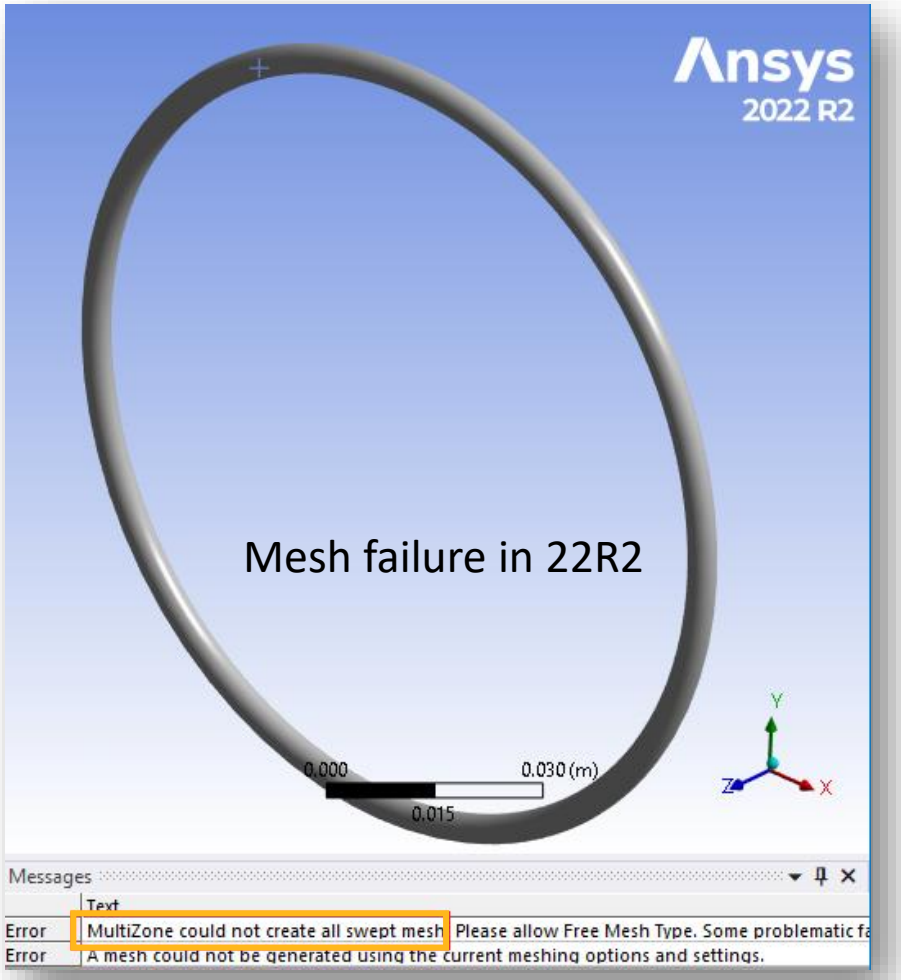


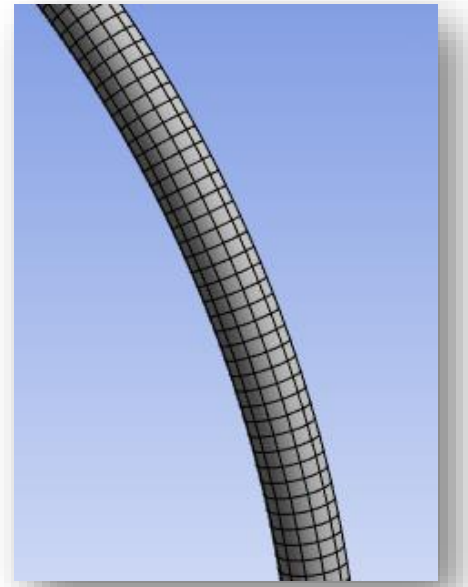
Default will be Yes but No is option for reverting to old behaviour for legacy scripts

©2022 ANSYS, Inc. / Confidential

Hex Meshing

Hex Meshing: Improved Primitive Handling for MultiZone

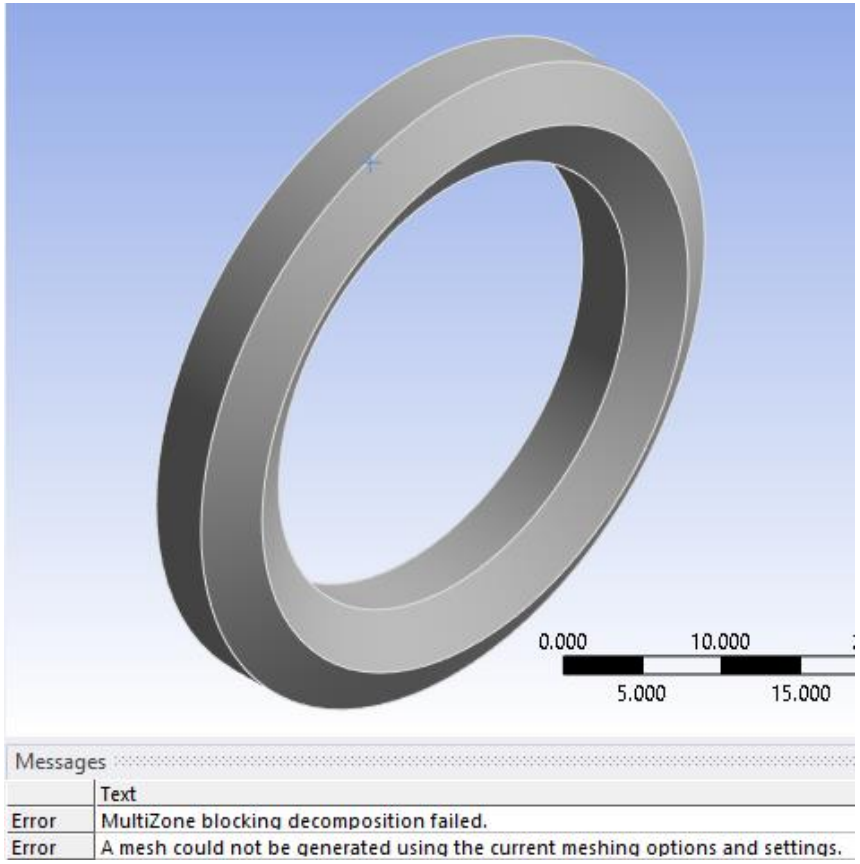




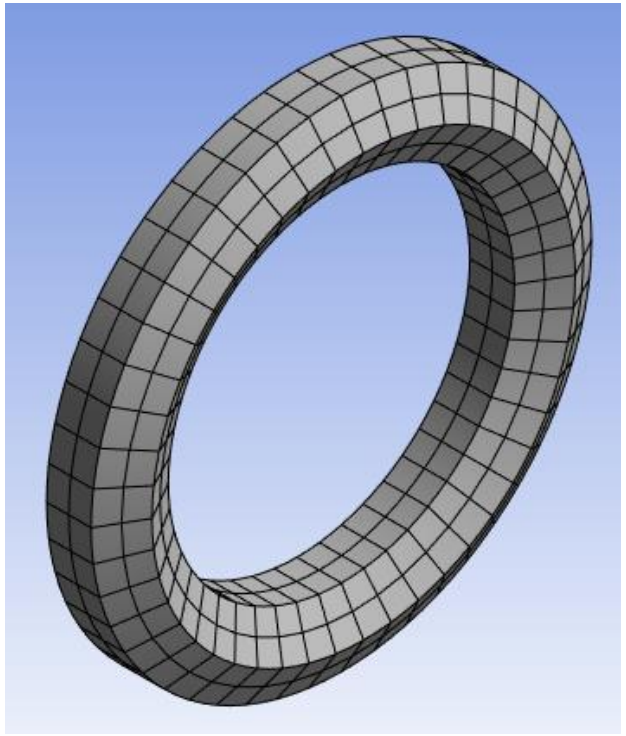
MultiZone is now able to mesh tori with no decomposition requirement

©2022 ANSYS, Inc. / Confidential

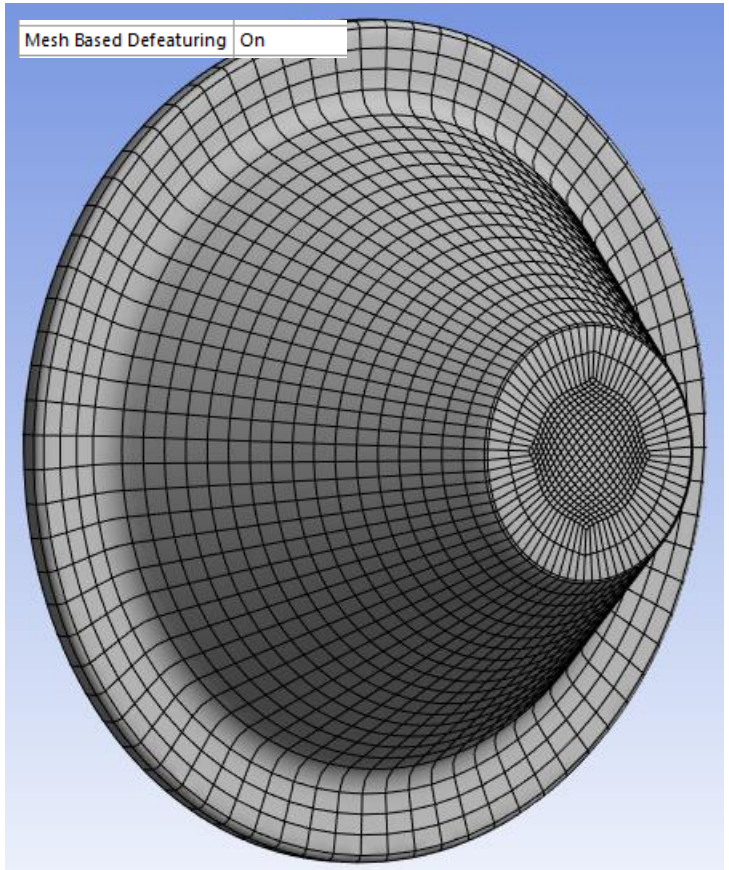
Hex Meshing: Improved Shape Handling for MultiZone



Mesh failure in 22.2



Mesh generated successfully in 23.1

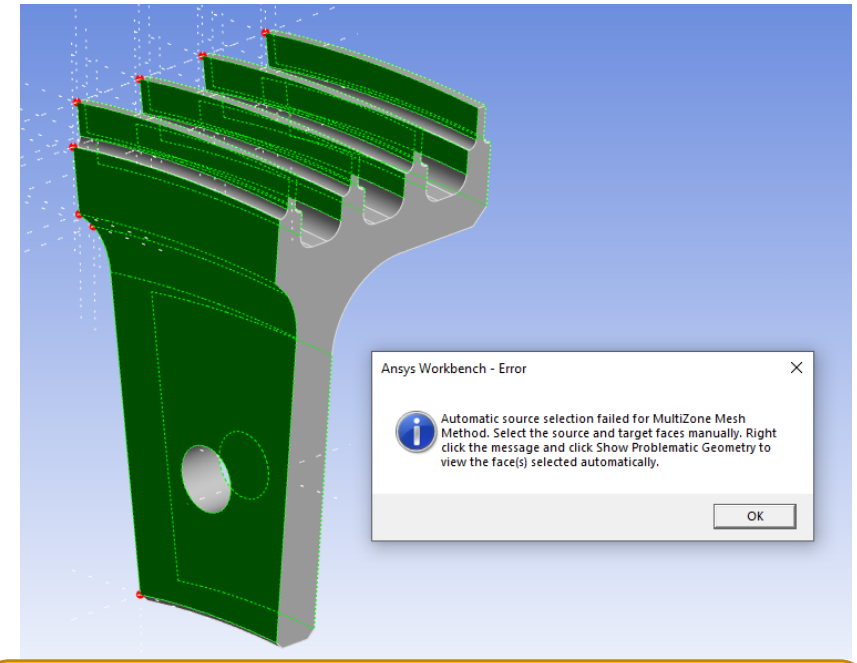


Defeaturing tolerance may need to be tweaked to mesh complex axisymmetric bodies

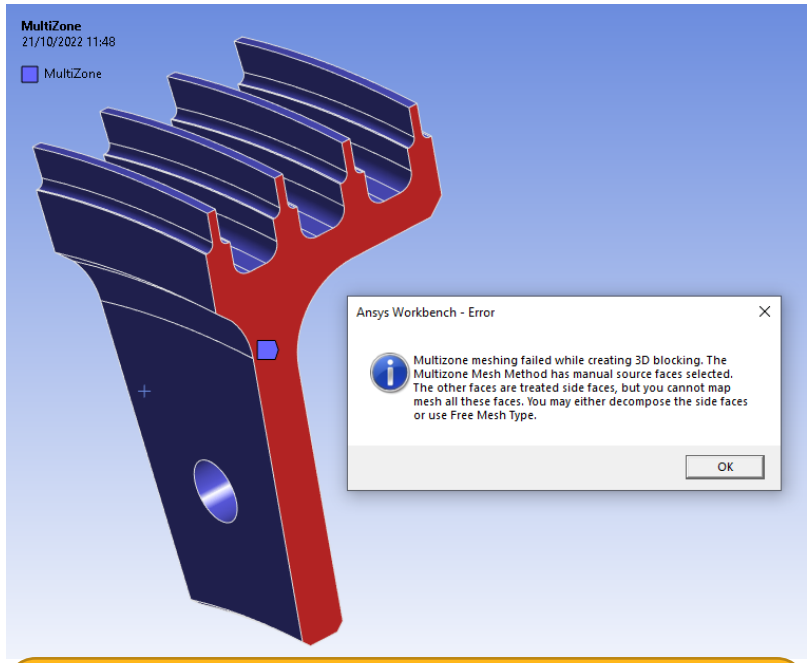
Multizone is now able to mesh axisymmetric bodies with no decomposition requirement

©2022 ANSYS, Inc. / Confidential

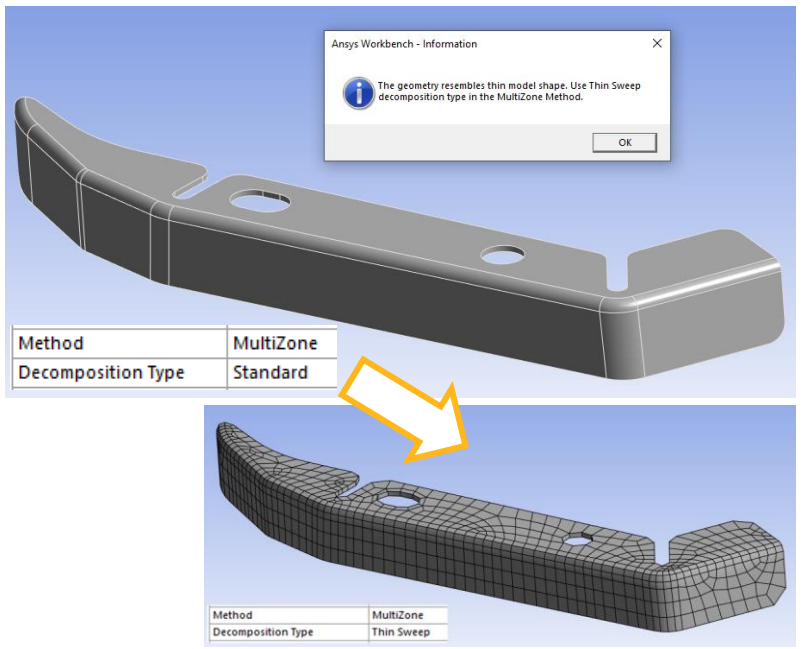
Hex Meshing: Improved error/info messaging for MultiZone



Automatic Source: If MZ fails the error message shows user which faces the automatic source selection chose and asks to manually select better options



If sweeping fails due to side faces not being “mappable” (topologically square) the offending face is highlighted as problematic geometry



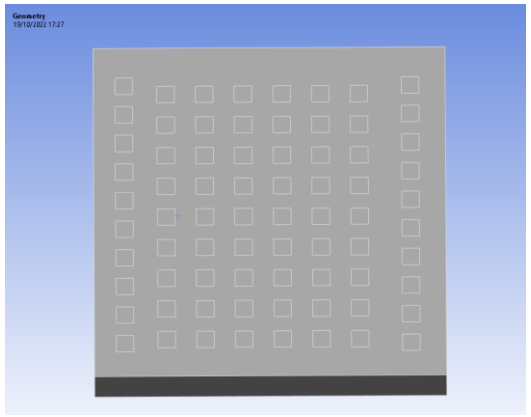
If MZ Meshing (Standard) fails but the body seems amenable to ThinSweep meshing the user is suggested to try this Decomposition approach

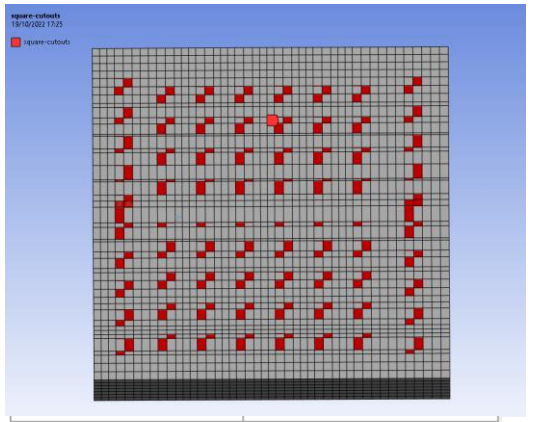
©2022 ANSYS, Inc. / Confidential

Hex Meshing - Body Fitted Cartesian Edge Smoothing

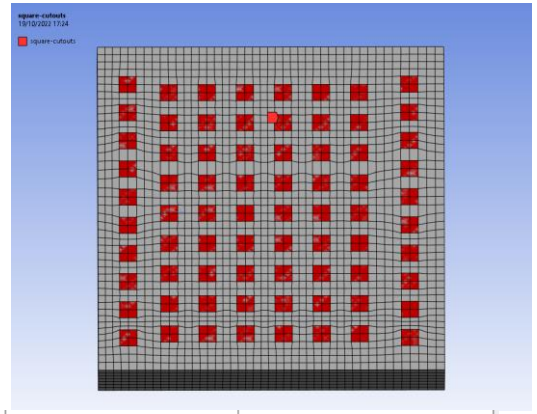
Details of "Body Fitted Cartesian" - Method

Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Method	Cartesian
Element Order	Use Global Setting
Type	Element Size
Element Size	1.0 mm
Spacing Option	Uniform
Advanced	
Projection Factor	0.98
Project in constant Z-Plane	No
Stretch Factor in X	1.0
Stretch Factor in Y	1.0
Stretch Factor in Z	1.0
Coordinate System	Global Coordinate System
Write ICEM CFD Files	No
Smooth Mesh Spacing	No





Smooth Mesh Spacing	No



Smooth Mesh Spacing	Yes

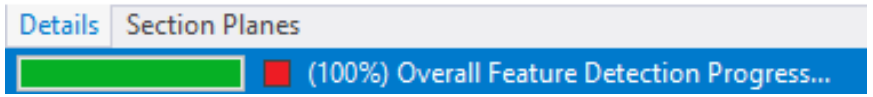
"Smooth Mesh Spacing" option uses edge smoothing to give more uniform mesh and proper protected topology (NS) for examples like this with mis-aligned, square imprints

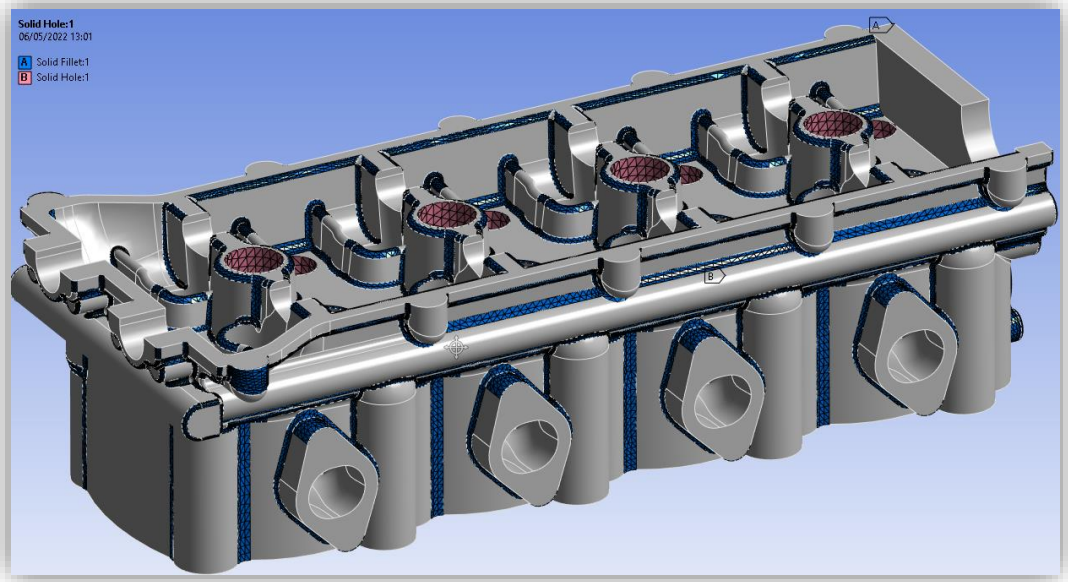
©2022 ANSYS, Inc. / Confidential

Feature Detection

Feature Detection

- Progress bar with interrupt has been added
- Ability to re-use generated NS groups for other Mesh Controls
- Performance improvements





Worksheet

Feature Detection

*Right click on the grid to add/delete a row.

	Name	Type	Criteria	Operator	Value	Angle	Min Bound	Max Bound	Mesh Treatment
1	Solid Fillet:1	Solid Fillet	Radius	Less Than or Equal	10	0	0	0	Mapped Meshing
2	Solid Hole:1	Solid Hole	Radius	Less Than or Equal	15	0	0	0	Mapped Meshing

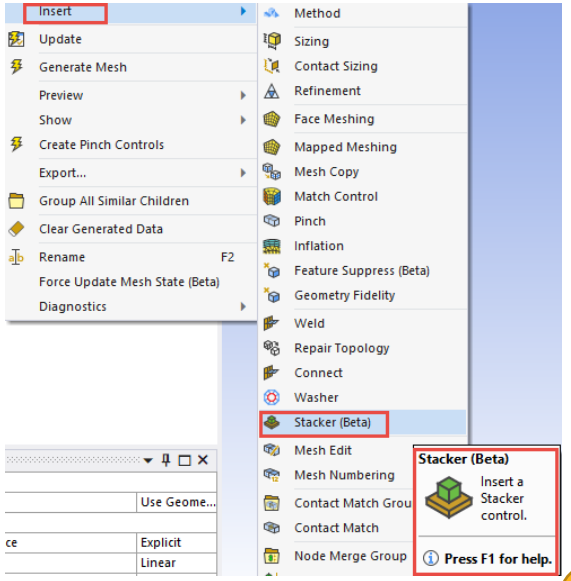
Add Feature Detection | Detect Features

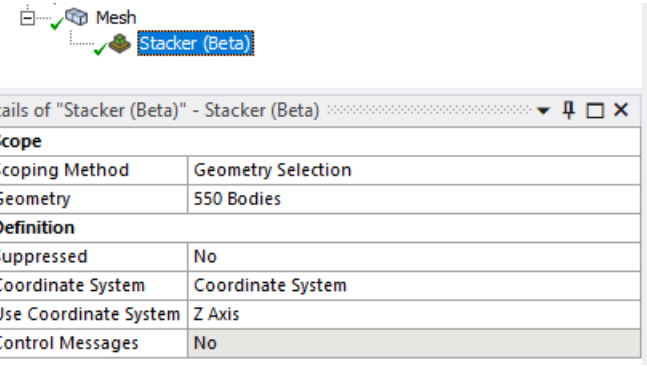
©2022 ANSYS, Inc. / Confidential

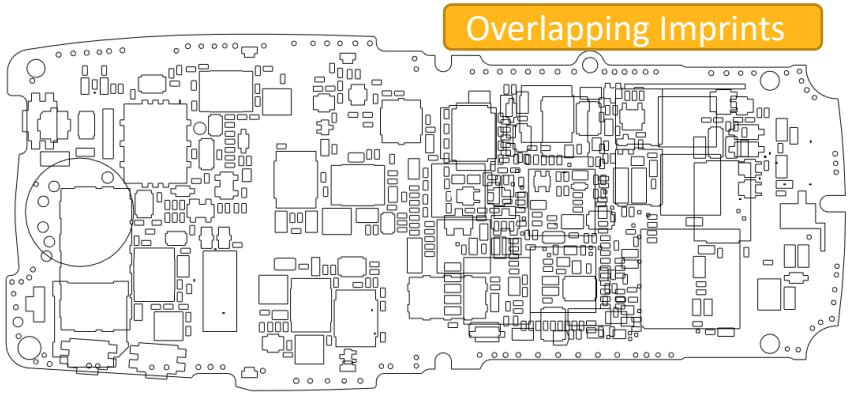
Mechanical Meshing
Beta Features

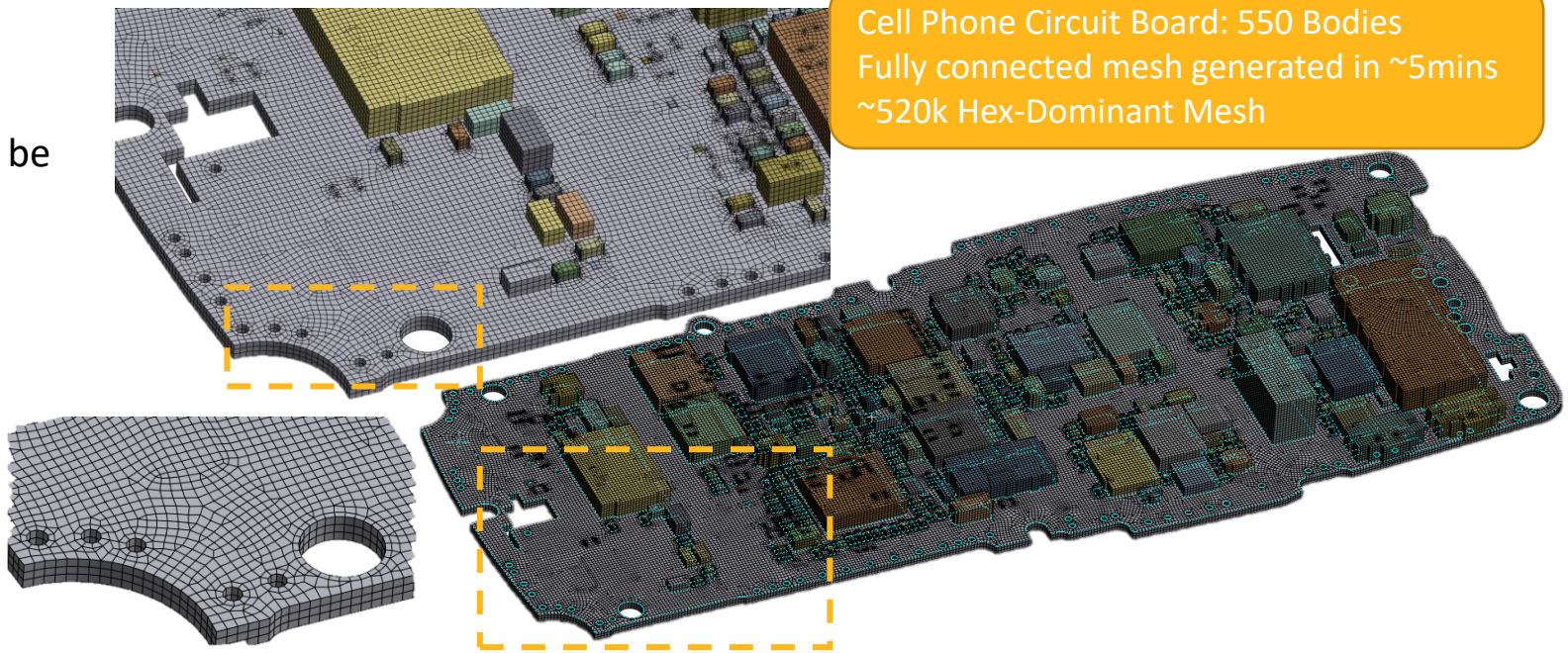
Stacker (2.5D) Meshing (Beta)

- New Stacker Mesh Method
  - Fast hex-dominant sweep meshing for 2.5D (normal-extruded) geometries
  - Supports overlapping imprints/loops between “stacked” bodies
  - Conformal mesh within a part
- 2023R1 Improvements
  - Several robustness issues addressed
  - Performance improvement – 7X speed-up in some test cases
  - Better sizing controls to get optimal mesh
  - Physics Preference driven sizing
- Note: Beta Features **and** Batch Connections must be enabled to see Stacker Control





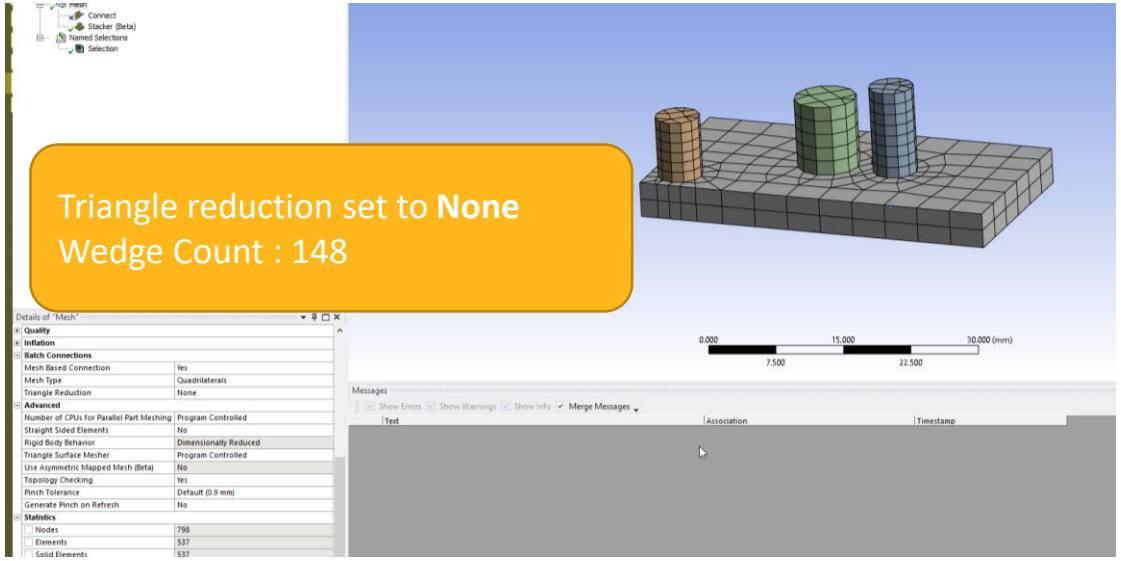




©2022 ANSYS, Inc. / Confidential

Stacker (Beta) Improvements

Triangle reduction set to **Aggressive**
Wedge Count: 84



Triangle reduction set to **None**
Wedge Count : 148

Stacker uses Batch Connections Surface Meshing Technology

Triangle reduction set to **Conservative**
Wedge Count : 108

©2022 ANSYS, Inc. / Confidential

Feature Detection Solid Logos - Beta

- Detection added for Solid Logos (Beta)
  - Feature Suppress treatment will remove detected features

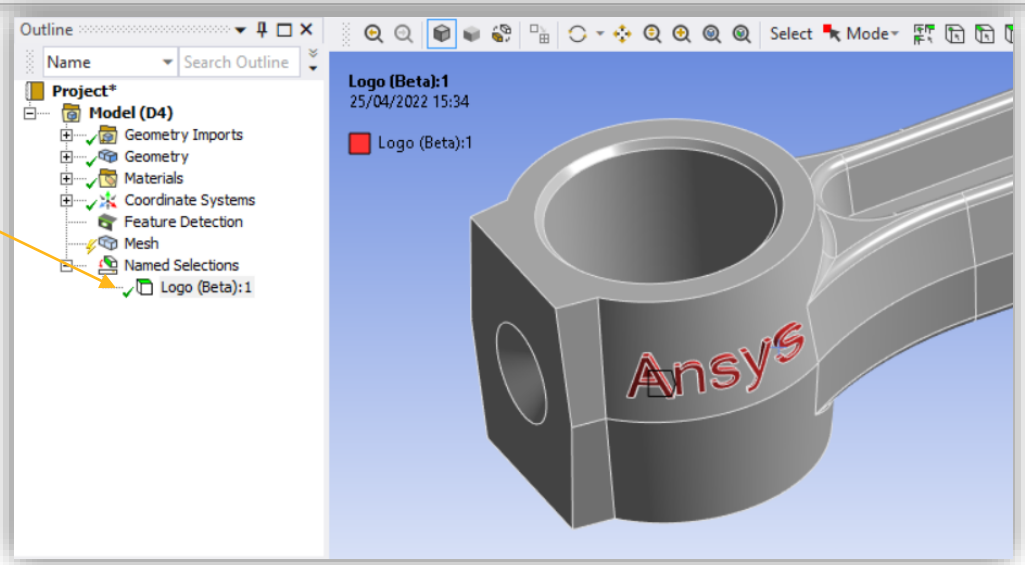
Worksheet

Feature Detection

*Right click on the grid to add/delete a row.

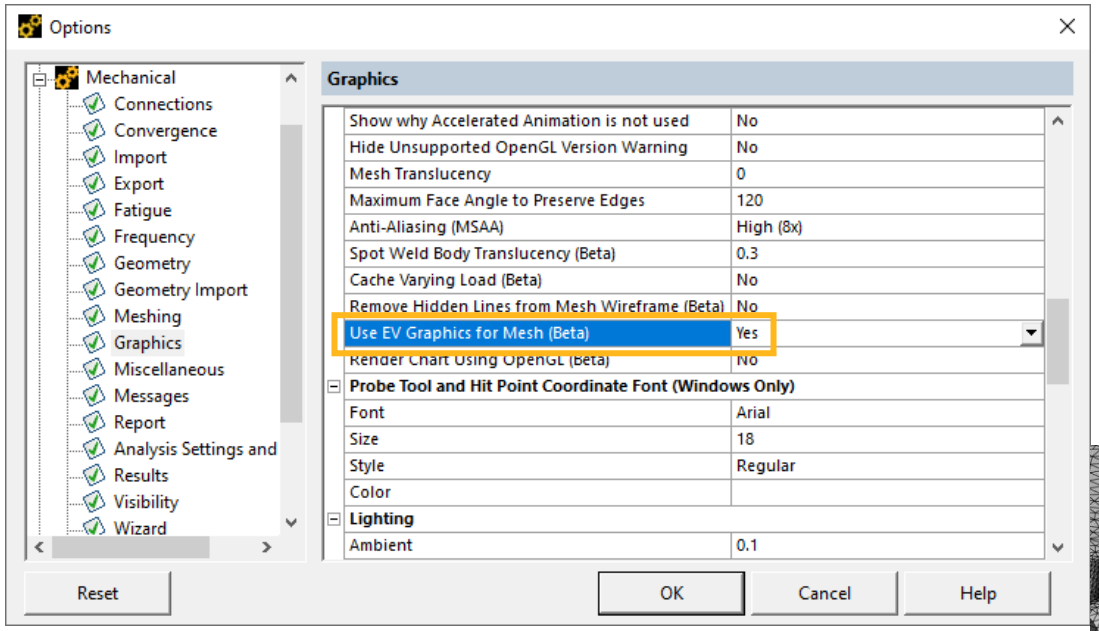
	Name	Type	Criteria	Operator	Value	Angle	Min Bound	Max Bound	Mesh Treatment
1	Logo (Beta):1	Solid Logos (Beta)	Height	Include range	0	0	0.4	5	None

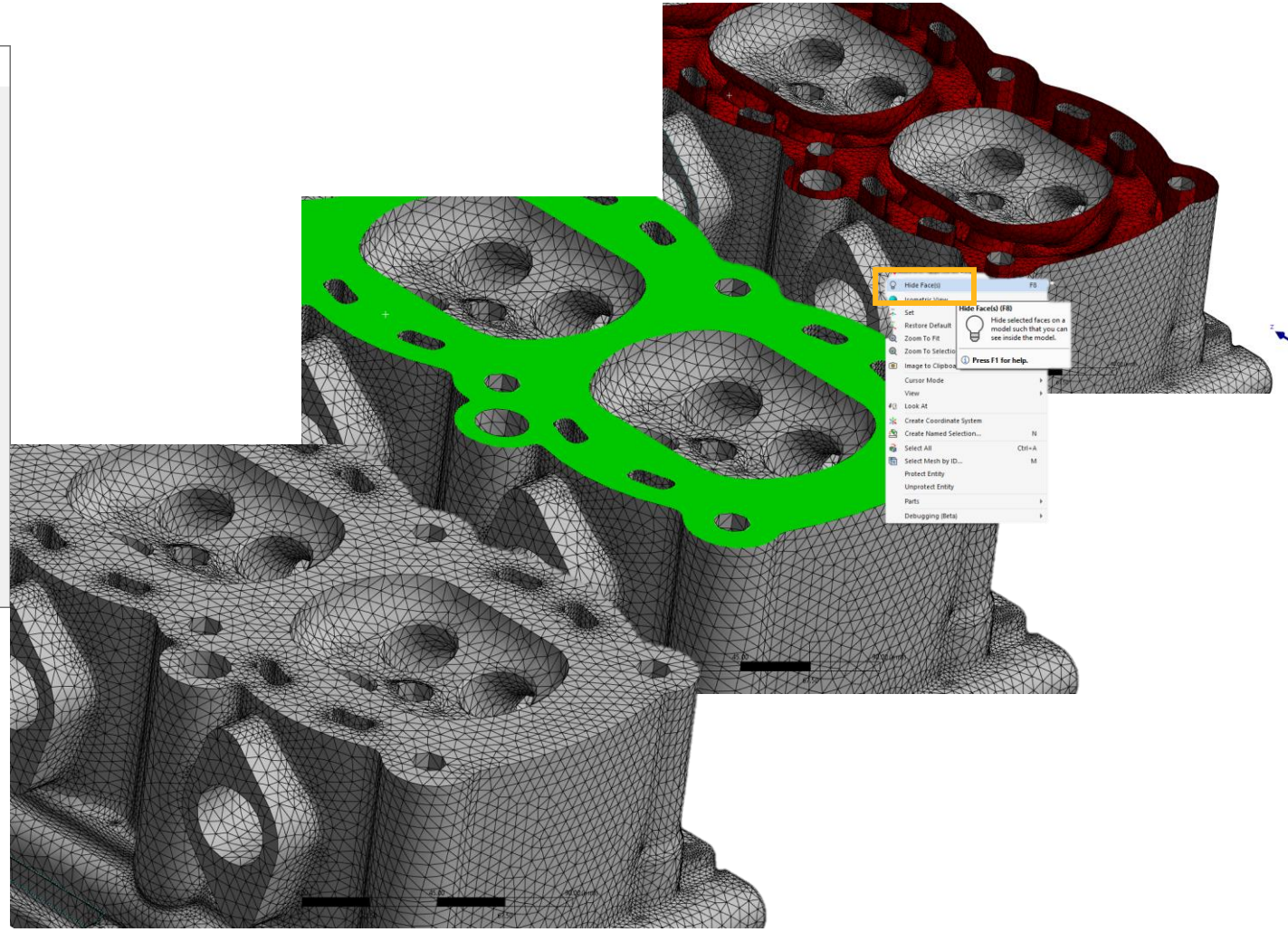
Add Feature Detection | Detect Features



©2022 ANSYS, Inc. / Confidential

Beta Option to allow Hide Mesh Faces



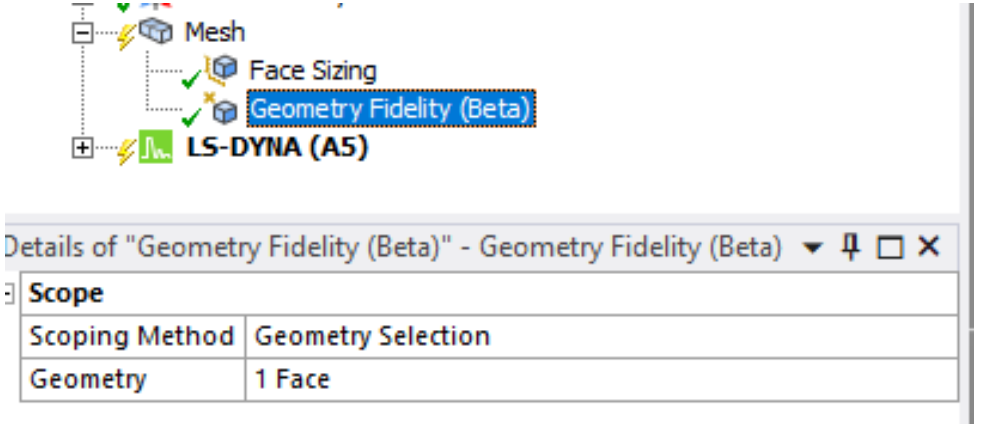


- A limitation exists for “Hide Faces” whereby the geometry is hidden but mesh does not get hidden
- This beta feature allows user to hide faces with mesh and see inside the shell mesh of a body to help diagnose meshing issues/needs

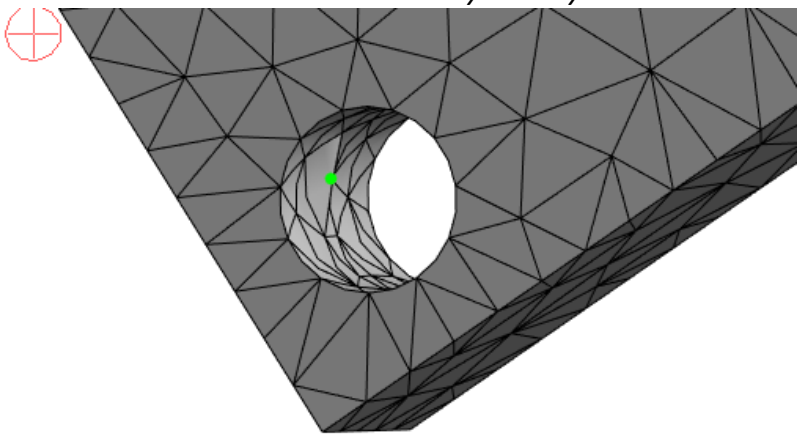
©2022 ANSYS, Inc. / Confidential

Geometry Fidelity Control (Beta)

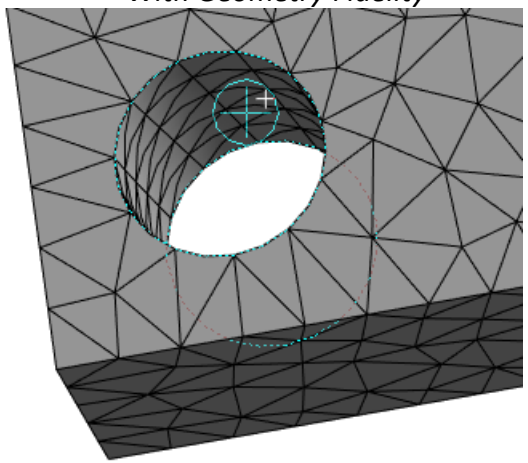
- New control added which is scopable to faces
- Suitable for scoping to bolt holes where user wants to ensure mid side nodes of 2nd order meshes are all on the geometry (e.g. for contact purposes)
  - Will impose map mesh on such surfaces
- In future releases, this control will be extended to other requirements where mesh nodes should remain on geometry



Without Geometry Fidelity

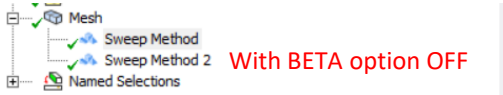


With Geometry Fidelity



©2022 ANSYS, Inc. / Confidential

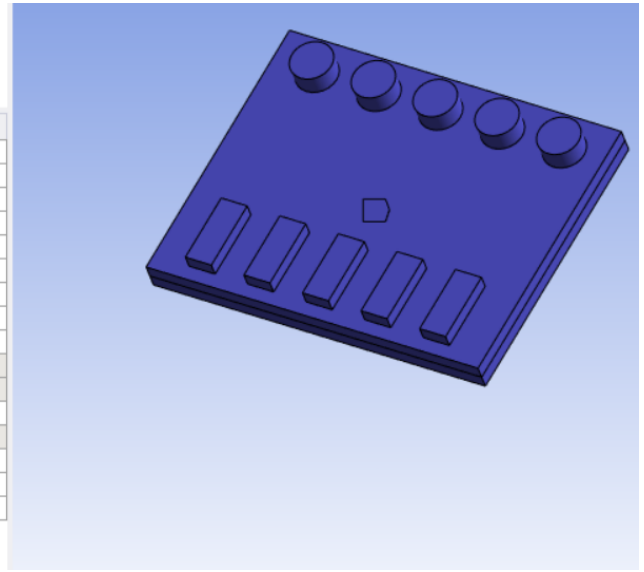
Enable # Divisions for Sweep for Multibodies w/ share topology → Automatic Thin (Beta)



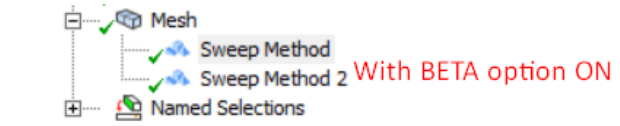
With BETA option OFF

Details of "Sweep Method" - Method

Scope	
Scoping Method	Geometry Selection
Geometry	16 Bodies
Definition	
Suppressed	No
Method	Sweep
Algorithm	Program Controlled
Element Order	Use Global Setting
Src/Trg Selection	Automatic Thin
Source Scoping Method	Program Controlled
Source	Program Controlled
Free Face Mesh Type	Quad/Tri
Sweep Num Divs	1
User Defined Criteria	Protect Internal Edges
Element Option	Solid
Constrain Boundary	No



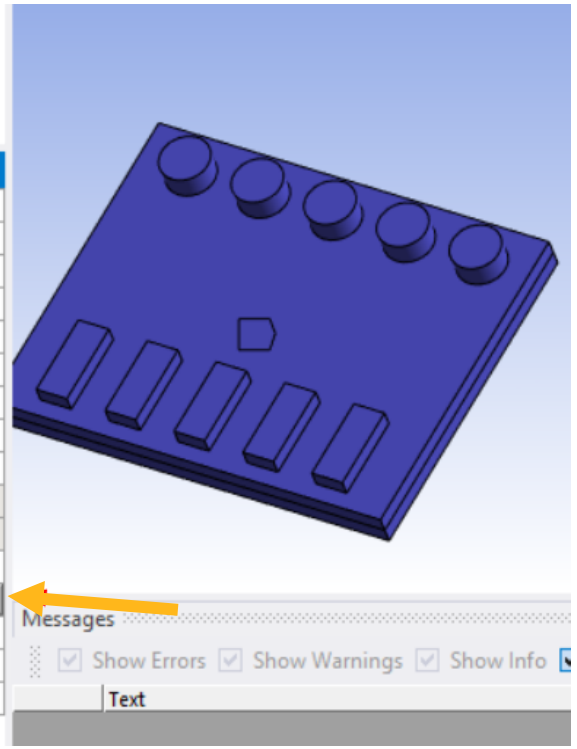
- A limitation exists for ability to set # divisions for "Automatic Thin" sweep (forced to 1) if body is part of multibody part with share topology (connected mesh)
- This feature allows user to set number of divisions for sweep



With BETA option ON

Details of "Sweep Method" - Method

Scope	
Scoping Method	Geometry Selection
Geometry	16 Bodies
Definition	
Suppressed	No
Method	Sweep
Algorithm	Program Controlled
Element Order	Use Global Setting
Src/Trg Selection	Automatic Thin
Source Scoping Method	Program Controlled
Source	Program Controlled
Free Face Mesh Type	Quad/Tri
Sweep Num Divs	3
User Defined Criteria	Protect Internal Edges
Element Option	Solid
Constrain Boundary	No



©2022 ANSYS, Inc. / Confidential

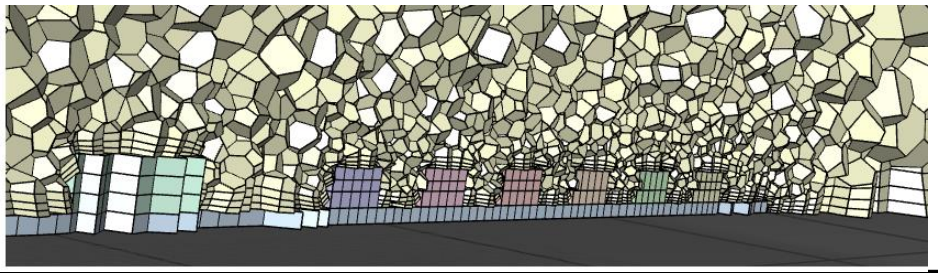
Fluent Meshing

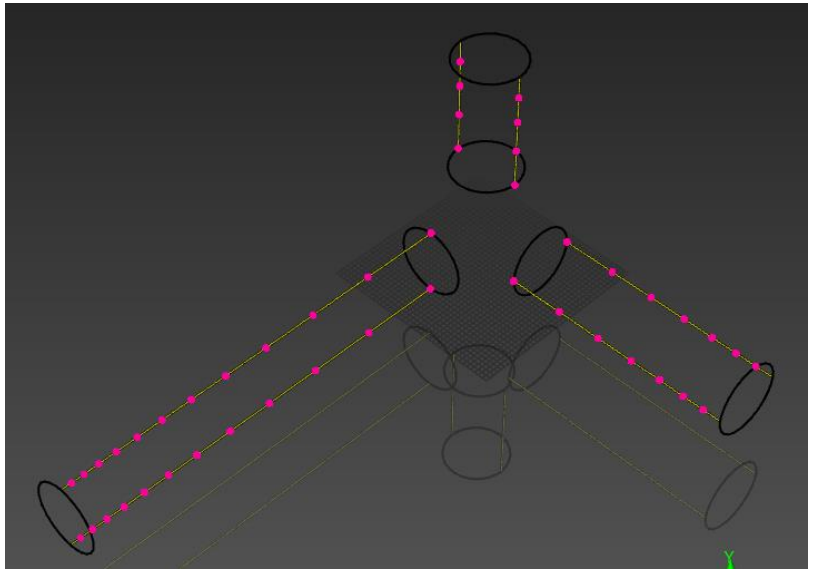
Watertight Meshing Workflow

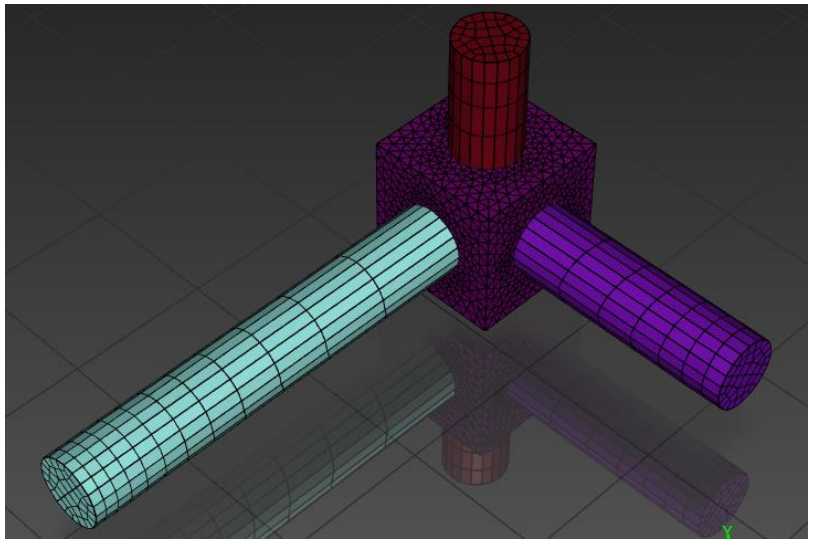
Ansys

Multizone Meshing Enhancements

- Edge-based sizing
  - Specify **Intervals**, **Size**, or **Smallest Height**
  - Single- and Bi-directional biasing
    - Specify **Growth Rate** or **Bias Factor**
  - Automatic edge-direction synchronization
  - Smart “Similar Edge” selection automatically selects all parallel edges of similar length
- “On-the-fly” choice of non-conformal connection to unstructured tet/hexcore
  - Can be used even if shared topology is present
  - Poly / poly-hexcore connection must be non-conformal



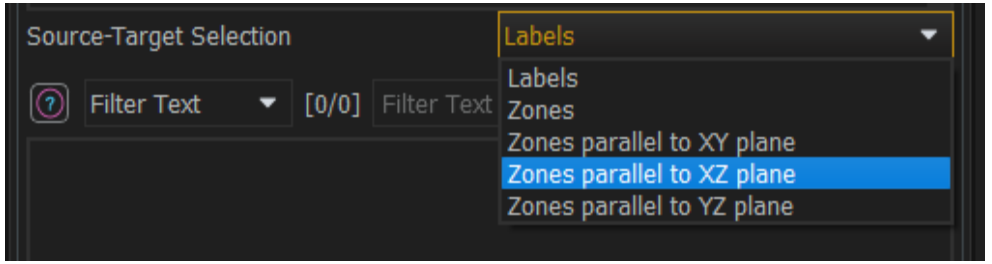




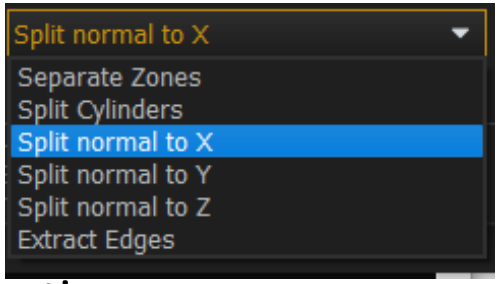
©2022 ANSYS, Inc. / Confidential

Additional Multizone selection features

- In Add Multizone Controls
  - Ability to select All source-target zones parallel to a global plane
    - Zones parallel to XY plane
    - Zones parallel to XZ plane
    - Zones parallel to YZ plane
  - Improved persistence with design changes



- In Manage Zones
  - Ability to split bodies normal to a global direction
    - Split normal to X
    - Split normal to Y
    - Split normal to Z
  - Split non-cylindrical shapes (e.g. heat exchanger tubes) in selected direction



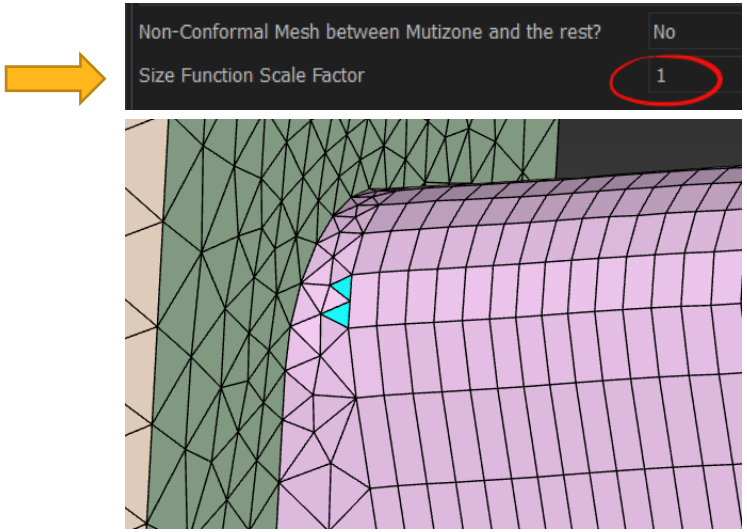
©2022 ANSYS, Inc. / Confidential

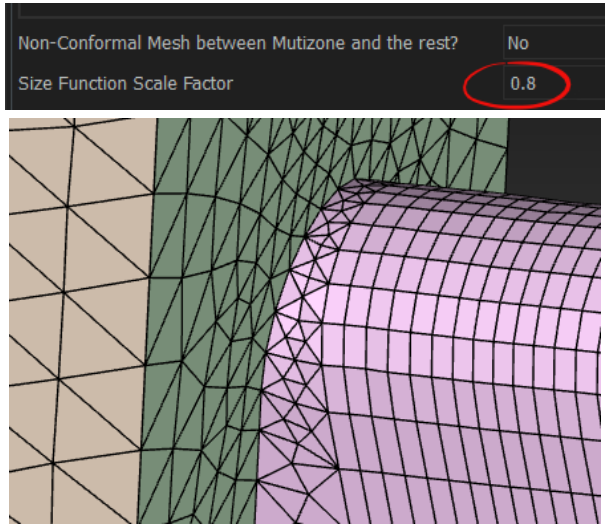
Multizone Size Function Scale Factor

- In some situations, the Multizone mesh is too coarse compared to the adjacent Surface mesh and the conformal connection is not completed, resulting in free faces

```
---- Warning--- The multizone regions (tube1 tube2) could not be conformally connected to the remaining surface mesh.
There are two work-arounds
---- Warning--- Either set the option 'Non-Conformal Mesh between Mutizone and the rest?' to 'yes', or reduce the
'MultiZone Size Function Scale Factor' to get a better size match
```

- A **Size Function Scale Factor** has been introduced that will scale the size field as seen by the multizone mesh
  - Located in the **Generate The Multizone Mesh** task

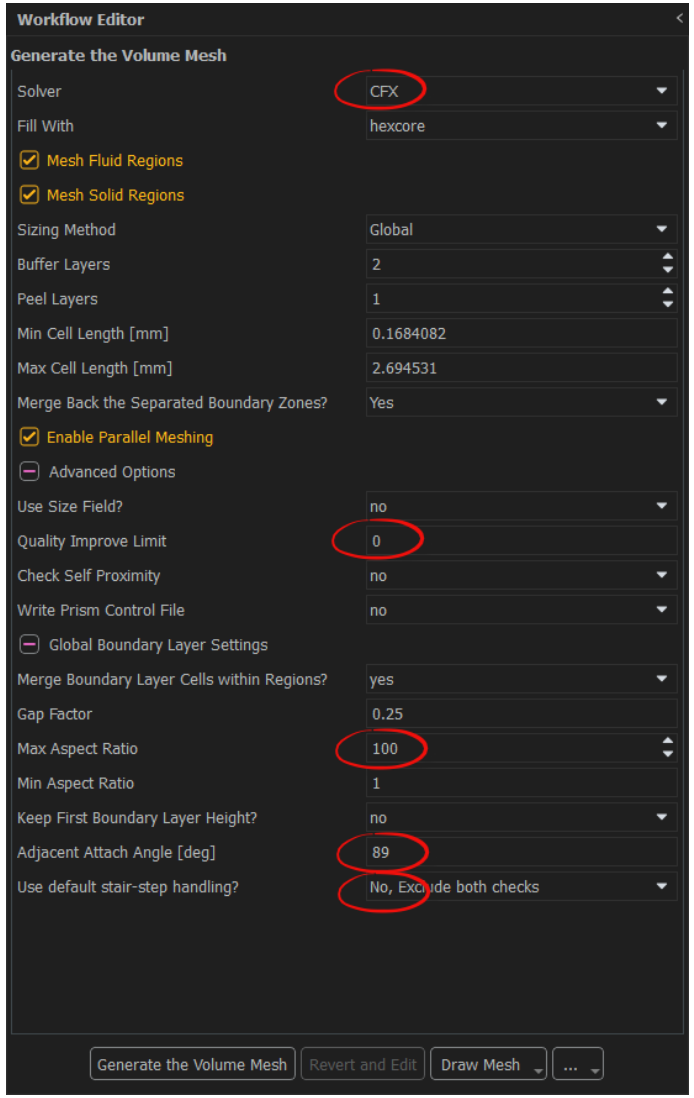




©2022 ANSYS, Inc. / Confidential

Solver Selection in Generate Volume Mesh

- Ability to choose Fluent or CFX as target solver
  - Avoids creating meshes that are incompatible and/or cause issues when read in CFX
    - Available volume fill types restricted:
      - Tetrahedral (default) and hexcore
    - Modifies **Global Boundary Layer Settings** defaults:
      - Max Aspect ratio 100
      - Adjacent angle 89
      - Exclude both stair-step checks
    - Advanced options hidden / modified
      - **Avoid 1:8 transition**
      - **Quality Limit** set to 0 – skips post-improve step that can cause negative volumes when converted in CFX



©2022 ANSYS, Inc. / Confidential

WTM Usability Enhancements - Region and Volume Mesh Options (1/2)

- Retain original name for dead regions
  - Voids or Dead regions are normally just named as "dead0", "dead1", etc.
  - But originally these regions are named after adjacent regions; fluid:1
  - New option in the Create Regions Task to retain the original names as suffixes

No - default

Region Name	Region Type
dead0	dead
dead1	dead
dead2	dead
dead3	dead
dead4	dead
dead5	dead
dead6	dead
dead7	dead
fluid1	fluid

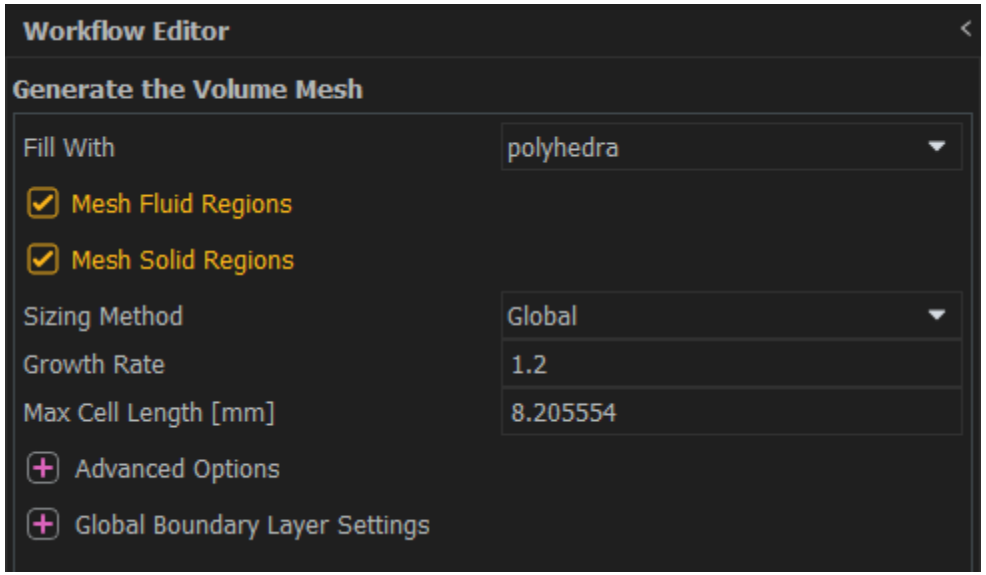
Yes

Region Name	Region Type
dead0-pre_e_nozzle-pre_e_nozzle:1	dead
dead1-zone20021-asm-20019-20019:1	dead
dead2-zone20021-asm-20019-20019:2	dead
dead3-zone20026_asm-20025-20025:1	dead
dead4-fluid:0	dead
dead5-fluid:3	dead
dead6-fluid:2	dead
dead7-fluid:1	dead
fluid1	fluid

©2022 ANSYS, Inc. / Confidential

WTM Usability enhancements - Region and Volume Mesh Options (2/2)

- Mesh Fluid or Solid Regions selectively
  - To complement the “Mesh Solid Regions” option in Generate the volume mesh task, a new option “Mesh Fluid Regions” has been added

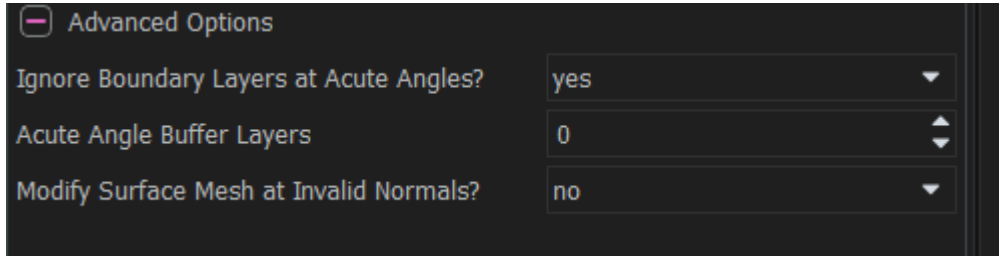


  - Allows independently creating fluid or solid regions of different types and save separately (FSI cases for example)
  - At least one must be enabled

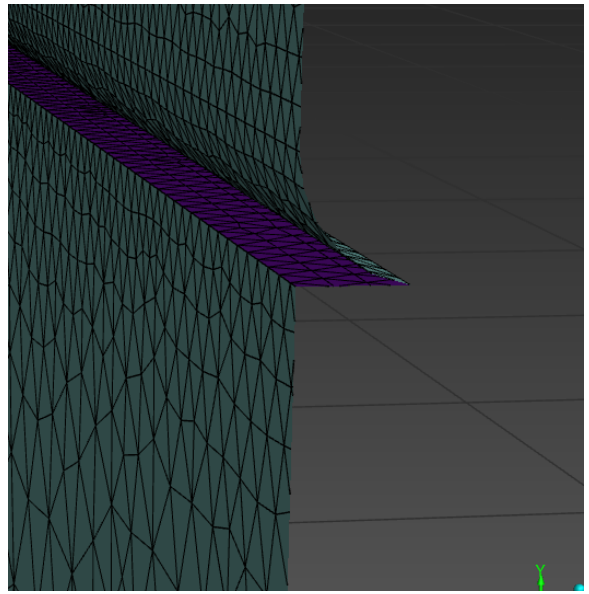
©2022 ANSYS, Inc. / Confidential

WTM Usability enhancements - Improve Boundary Layer settings

- Add Buffer layers to boundary-layer-ignore at acute angles
  - A new advanced option has been added in Add boundary layer task



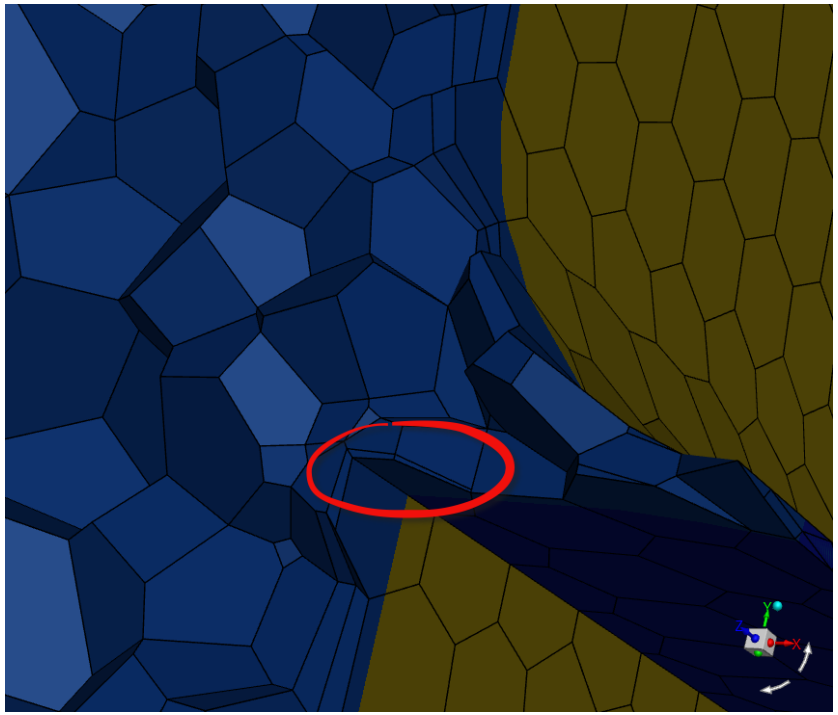
  - Increasing the Acute Angle Buffer Layers value increases the number of faces for which boundary layer will be ignored at acute angles



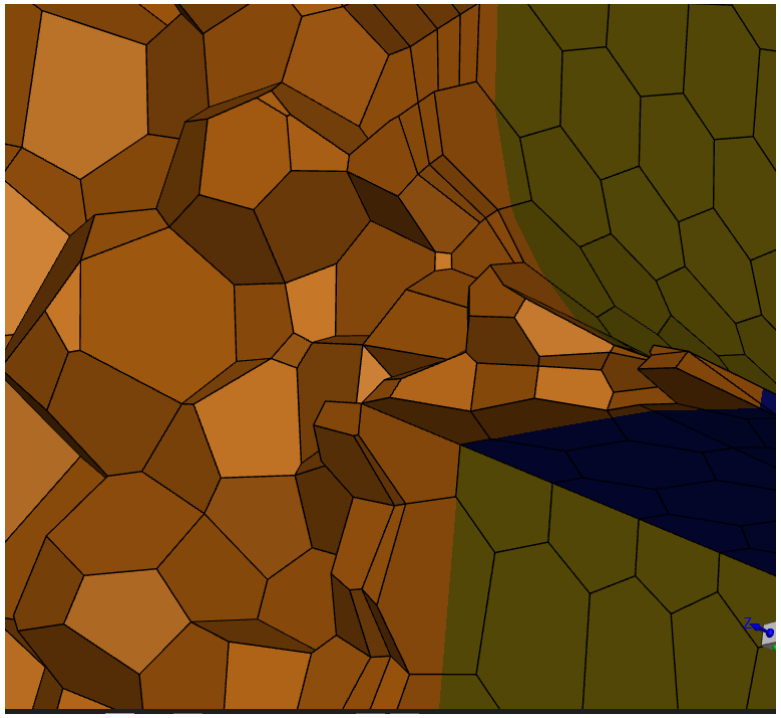
©2022 ANSYS, Inc. / Confidential

WTM Usability enhancements - Improve Boundary Layer settings

- Example
  - The user wants to avoid that boundary layers are grown in the fillet area



Example with Acute Angle Buffer layer = 0

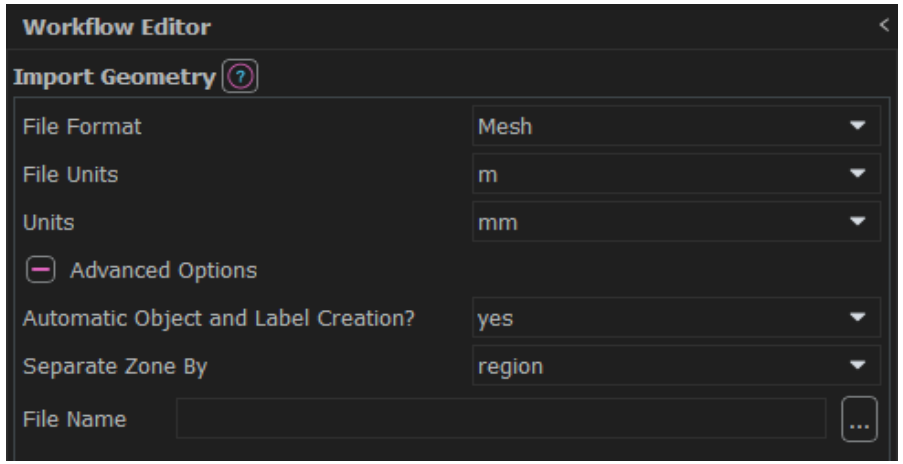


Example with Acute Angle Buffer layer = 1

©2022 ANSYS, Inc. / Confidential

WTM Usability enhancements - Additional import Options

- Simplified Mesh import
  - A new advanced option; “Automatic Object and Label Creation?”, has been added in Import Geometry, when the File Format is set to Mesh

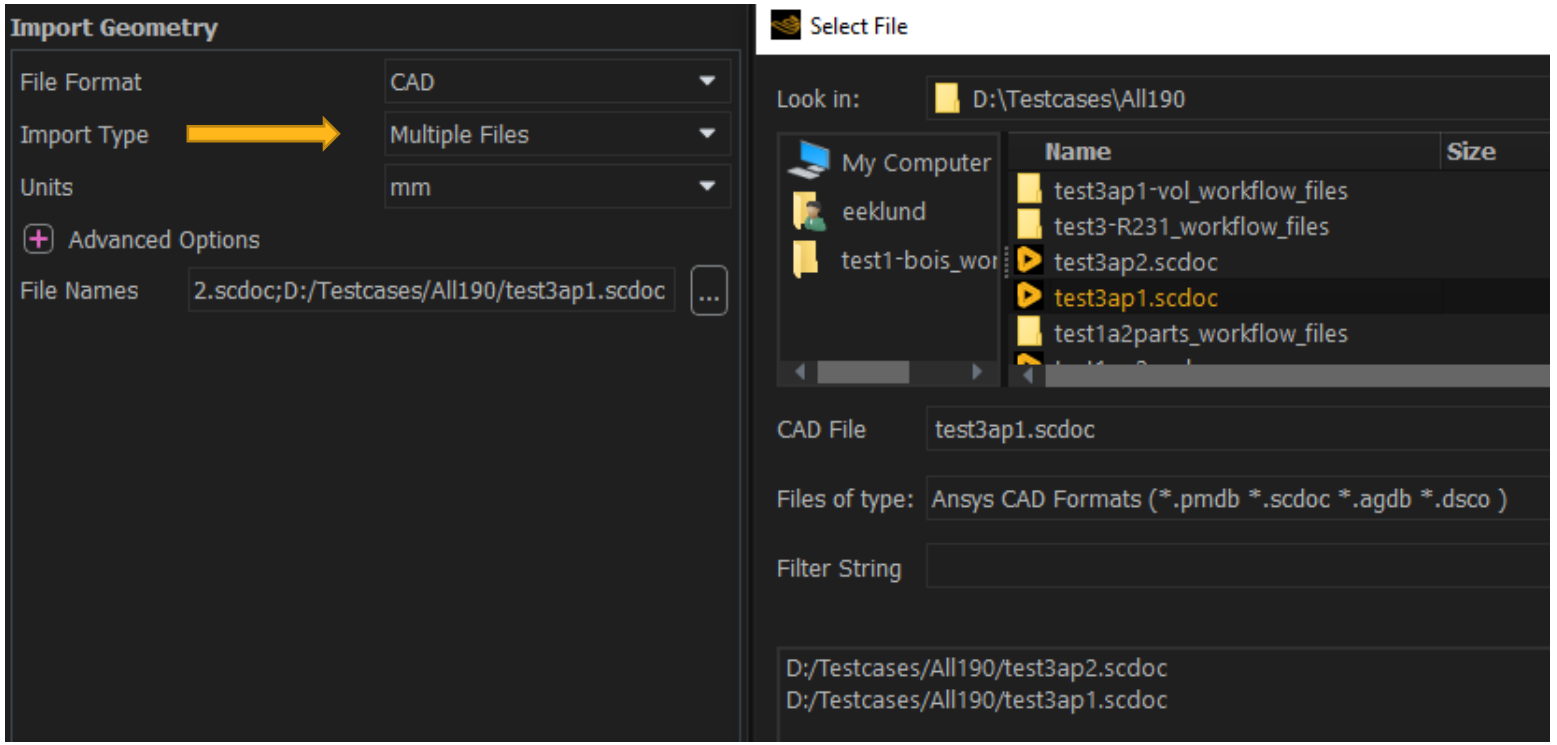


  - By default, the option is set to “yes”.
  - If this option is set to “no”:
    - No Labels are created
    - A single mesh object is created using all zones
      - Exception: Only zones including the string “*boi*” are excluded from the object creation
  - For very large cases, this will dramatically increase the speed of mesh import

©2022 ANSYS, Inc. / Confidential

WTM Usability enhancements - Additional import Options

- Multiple CAD import
  - A new option; “Import Type”, has been added in Import Geometry, when the File Format is set to CAD
  - By default, the option is set to “Single File”
  - If this option is set to “Multiple Files”:
    - The File selection panel changes to multiple selections
    - Files selected must be
      - In the same Directory
      - Of the same CAD format

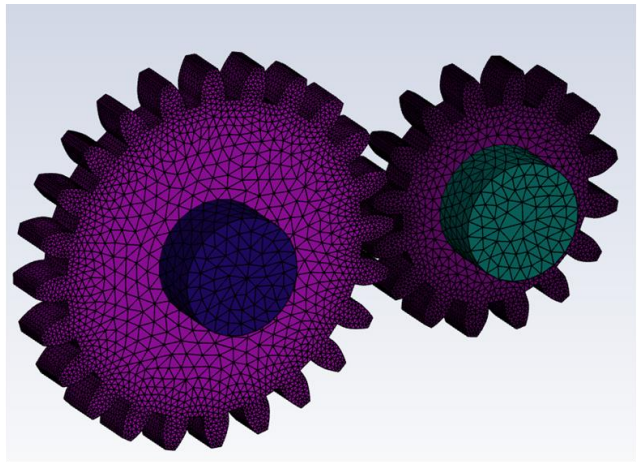


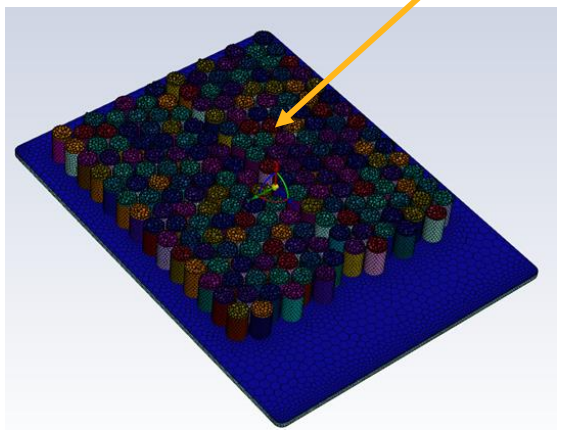
©2022 ANSYS, Inc. / Confidential

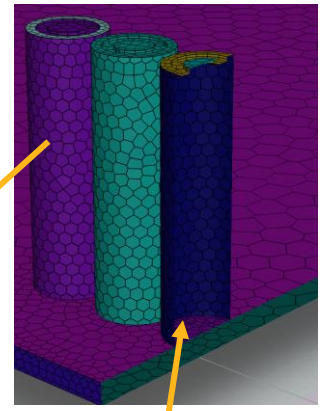
Fault-Tolerant Meshing

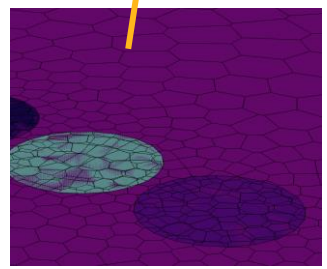
FTM: CFD Surface Mesh / Conformal Mesh

- New extraction option in Update Region Settings: "cfd surface mesh"
  - Uses new surface meshing technology offering higher performance /robustness on some challenging cases
- Optional geometry projection during extraction; option in Generate Surface Mesh
  - Project on Geometry of the CFD Surface Mesh Objects [Disabled by default]









Update Region Settings

Main Region

Displayed Regions: All Regions

Filter: Filter Text

Name	Type	Extraction Method	Volume Fill	Lea
fluid_driven_gear	solid	cfd surface mesh	poly	
fluid_driving_gear	solid	cfd surface mesh	poly	

Non-conformal meshes generated where overlapping faces are not shared

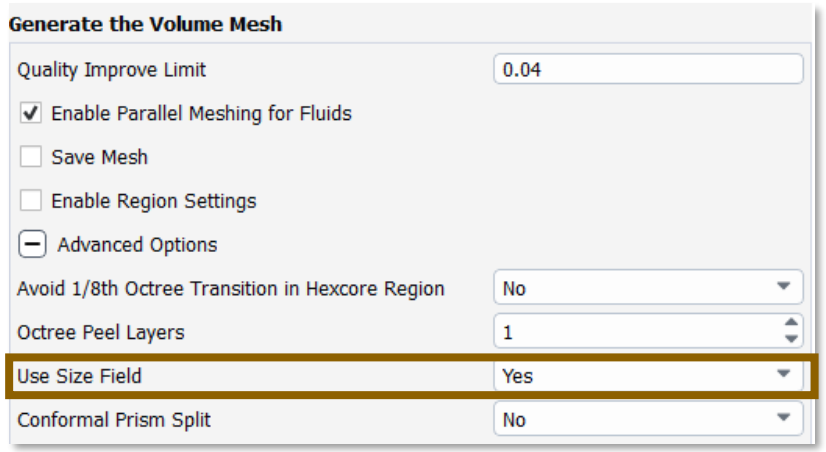
Conformal meshes created where faces are shared

©2022 ANSYS, Inc. / Confidential

Fill Without Size Field

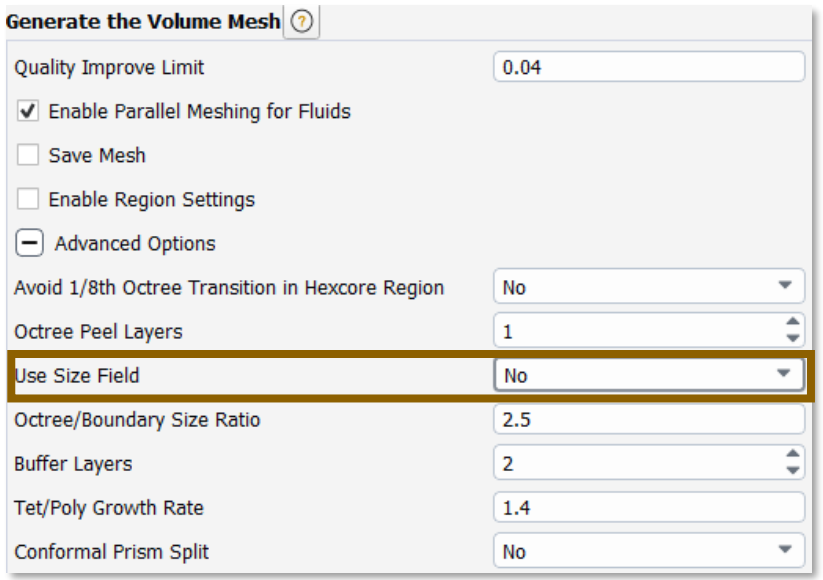
- When volume meshing in parallel using size fields, total memory is increased by duplication of size fields
- **Use Size Field** = **No** uses specified parameters to govern interior mesh size
  - **Octree Peel Layers**
  - **Octree/Boundary Size Ratio**
  - **Buffer Layers**
  - **Tet/Poly Growth Rate**

Small battery (no solids)
20 cores on single node (windows)



Total cell count: 1417957
14.7 minutes

```
| Virtual Mem Usage (GB)     |
| Current       Peak         |
| 44.7191       51.3932      |
```



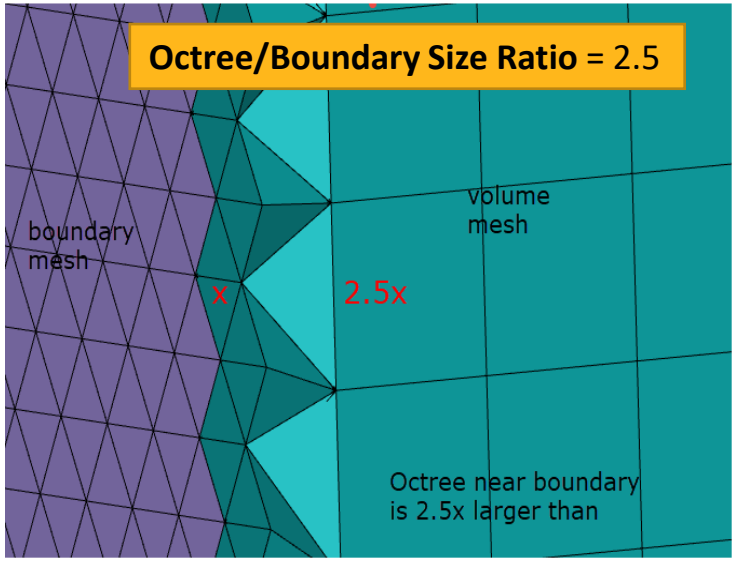
Total cell count: 1494872
11.5 minutes

```
| Virtual Mem Usage (GB)     |
| Current       Peak         |
| 13.1558       19.2074      |
```

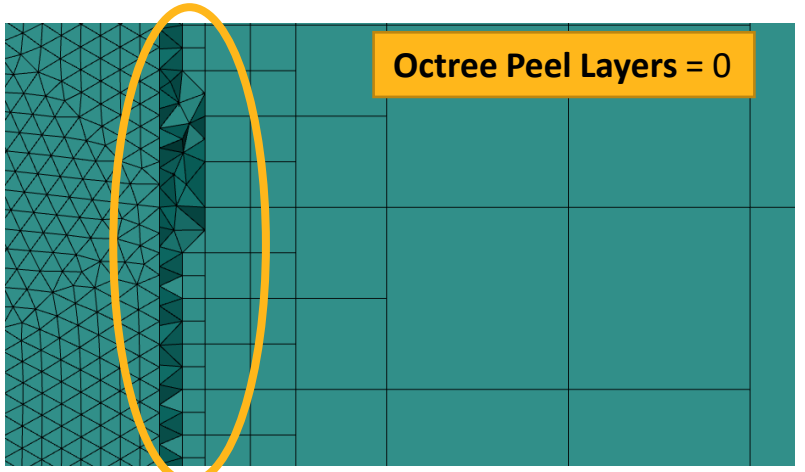

©2022 ANSYS, Inc. / Confidential

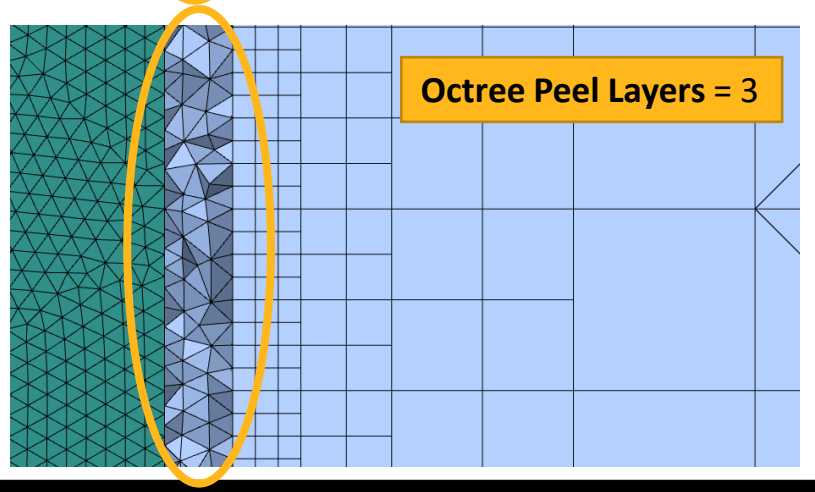
Fill Without Size Field Parameters

Octree/Boundary Size Ratio



Octree Peel Layers

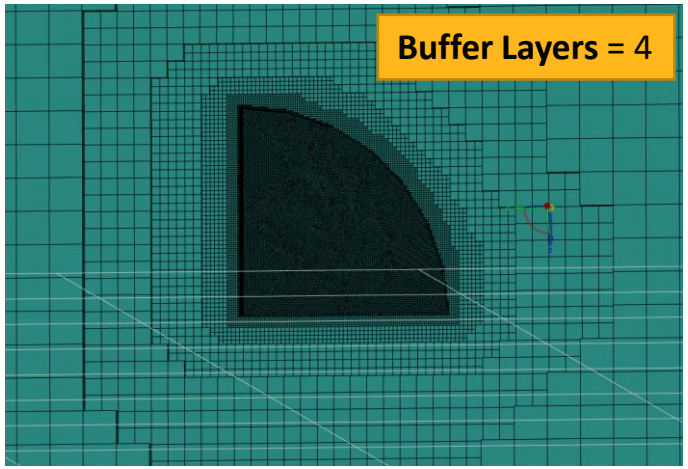


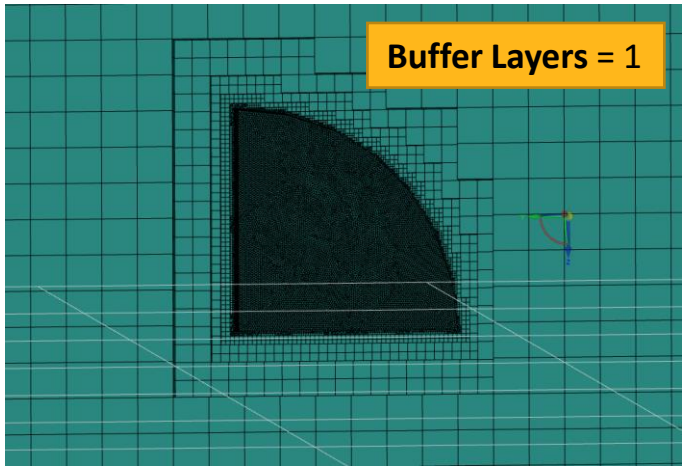


©2022 ANSYS, Inc. / Confidential

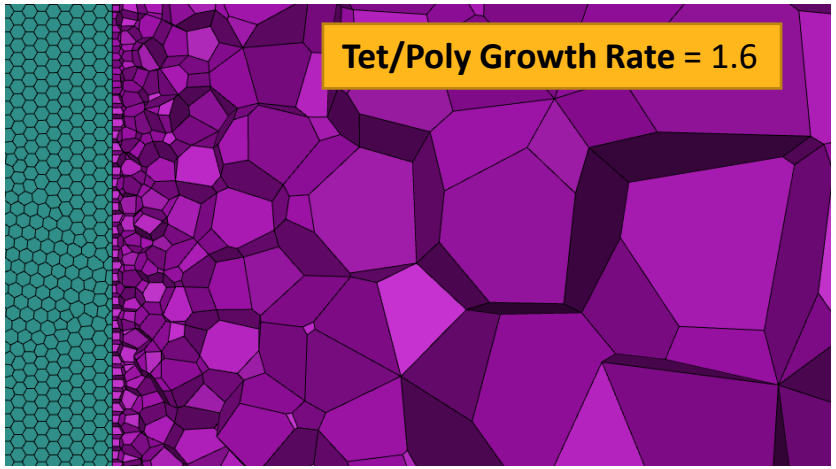
Fill Without Size Field Parameters

Buffer Layers





Tet/Poly Growth Rate

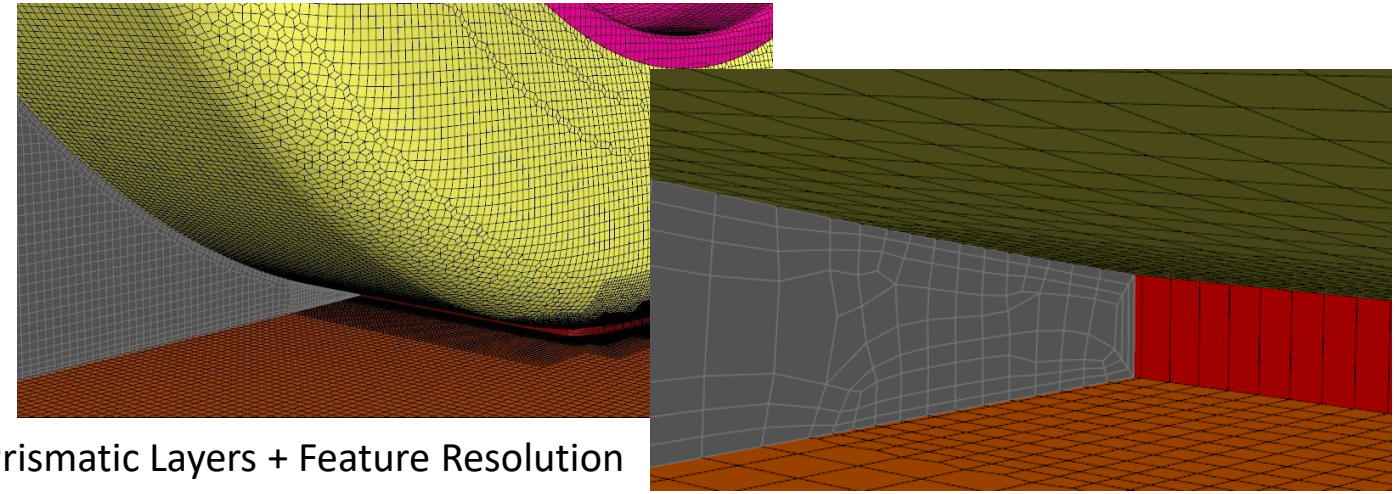


©2022 ANSYS, Inc. / Confidential

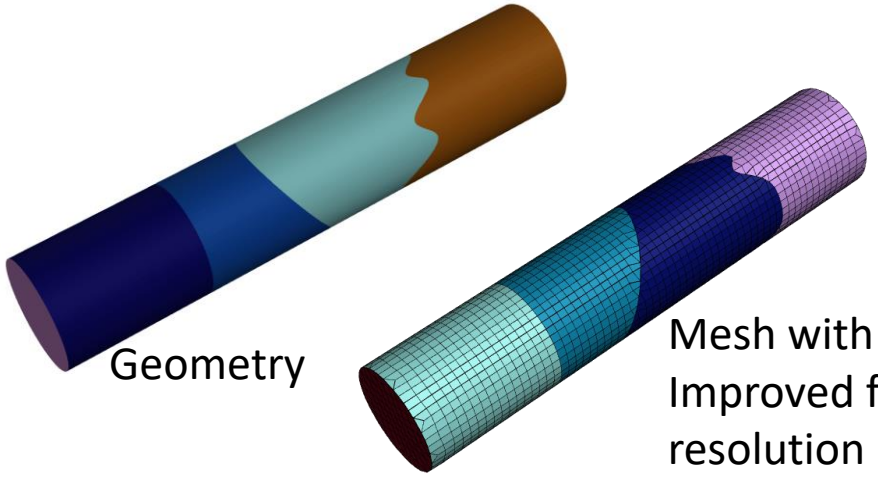
Rapid Octree

Rapid Octree Improvements

- Core Functionality
  - *Prismatic Layers switched to full release in 2023 R1*
    - Projection angle correction for Prismatic Layers (significantly reduced bad cell count)
    - Improved splitting parameter computation
    - Additional prism bunching modes (e.g. First Layer Aspect Ratio)
  - *Improved Geometry Resolution switched to full release in 2023 R1*
    - Improved Mesh Optimization Scheme
    - Code Runtime optimization



Prismatic Layers + Feature Resolution



Geometry

Mesh with Improved feature resolution

©2022 ANSYS, Inc. / Confidential

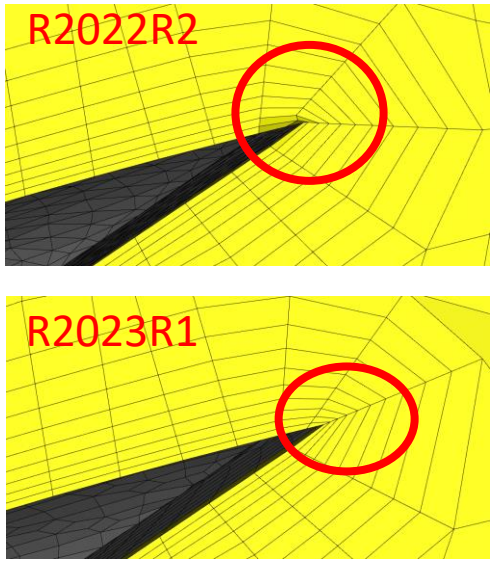
Rapid Octree: Projection Angle Correction For Prisms

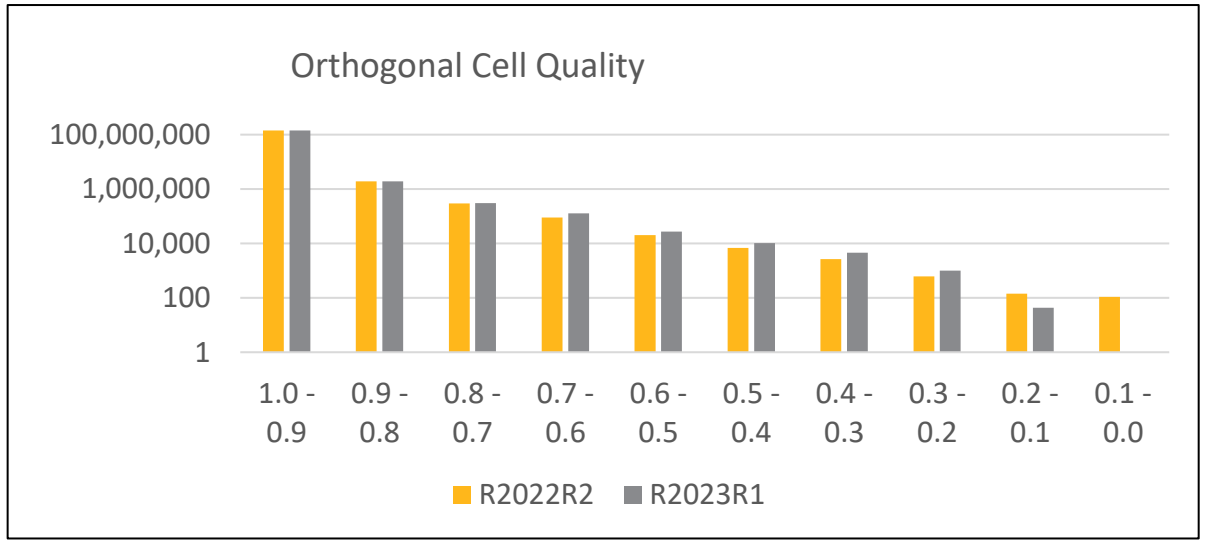
- Example of customer benchmark case
  - 3 Prismatic Layers
  - Non-Uniform surface sizing + Refinement Regions
  - Improved Geometry Resolution Active
  - 142M Cells

- 2022 R2
  - Minimum Orthogonal Quality 2.51753e-02
  - After RO, 7 bad quality cells (< 0.01) needed post-fix with auto-node-move (ANM)
- 2023 R1
  - Minimum Orthogonal Quality 1.17547e-01
  - No post-fix by ANM required

➢ *Significant reduction in bad cell count*

Angle Correction in Prismatic Layers

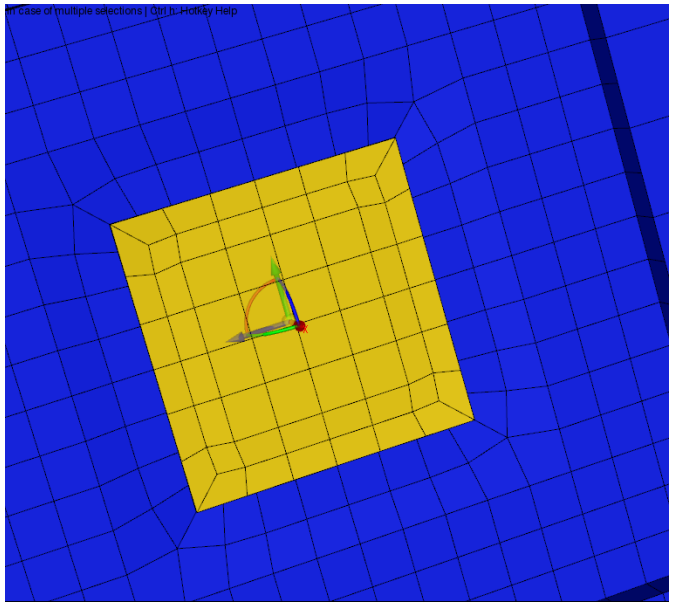




©2022 ANSYS, Inc. / Confidential

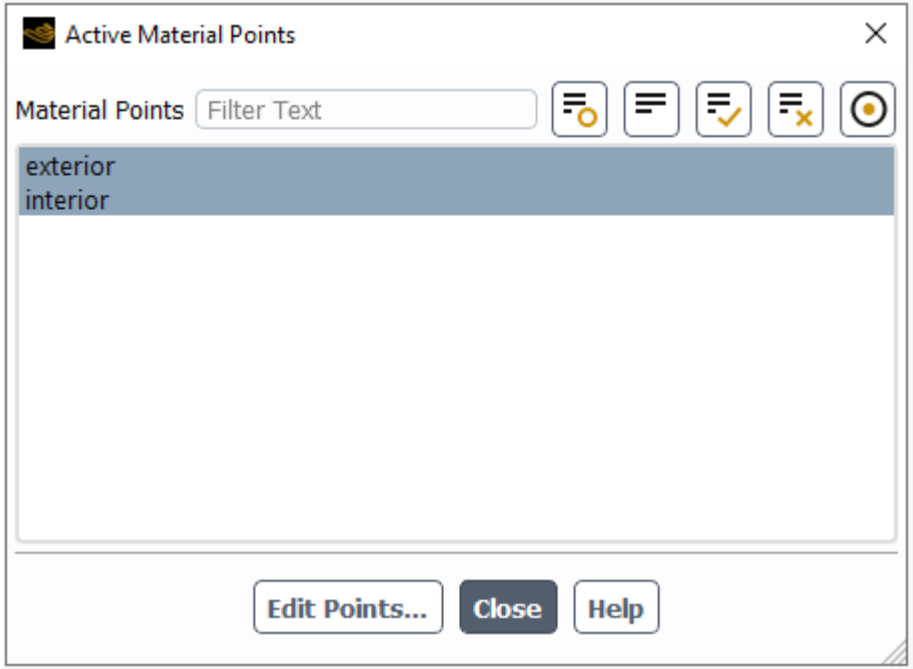
Rapid Octree – Generic Material Point Option

- Specify Material Points to enable multivolume meshing of dirty CAD input
- Requires large leakages to be closed before meshing
- Results in non-conformal multivolume



New Volume Option

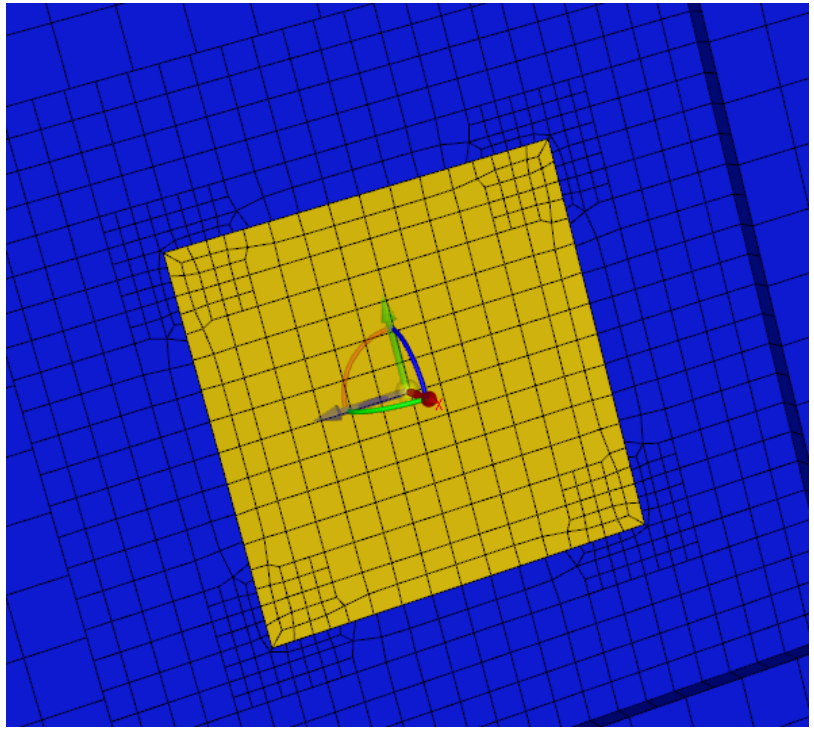
Select MPT Panel



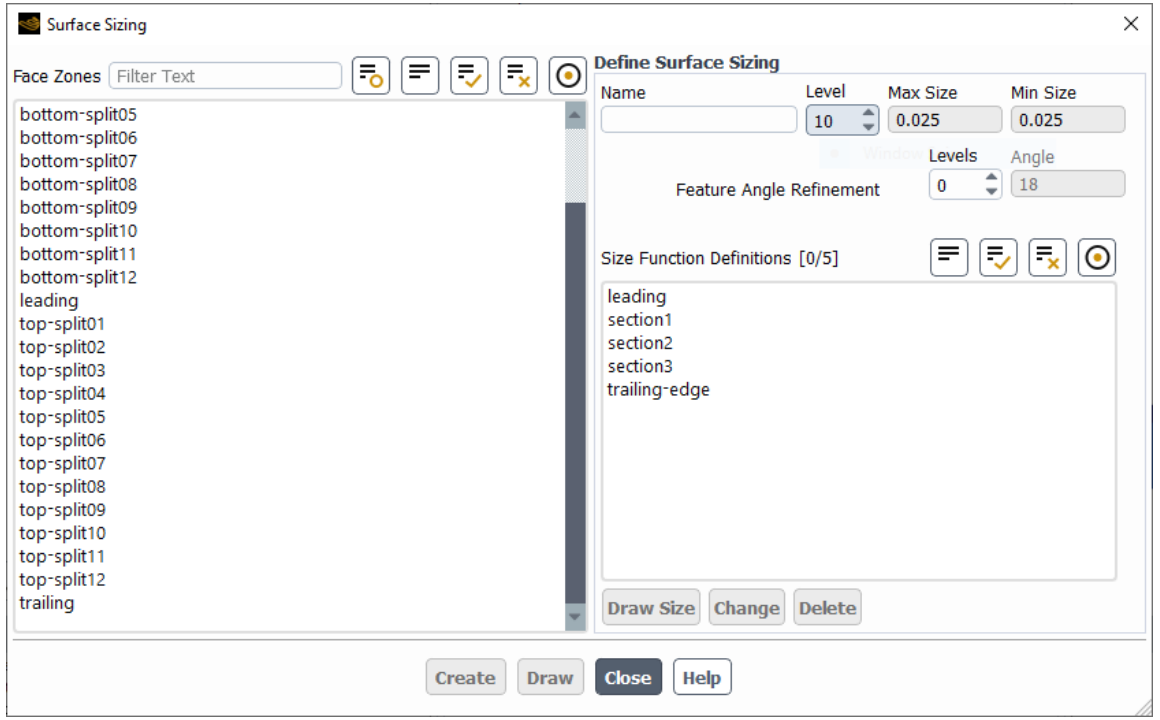
©2022 ANSYS, Inc. / Confidential

Rapid Octree – Angular Refinements for Boundary Sizing

- R2022R2: Only a global value can be given for angular refinements
- Enables finer adjustments of the mesh density



Extended Boundary sizing Panel

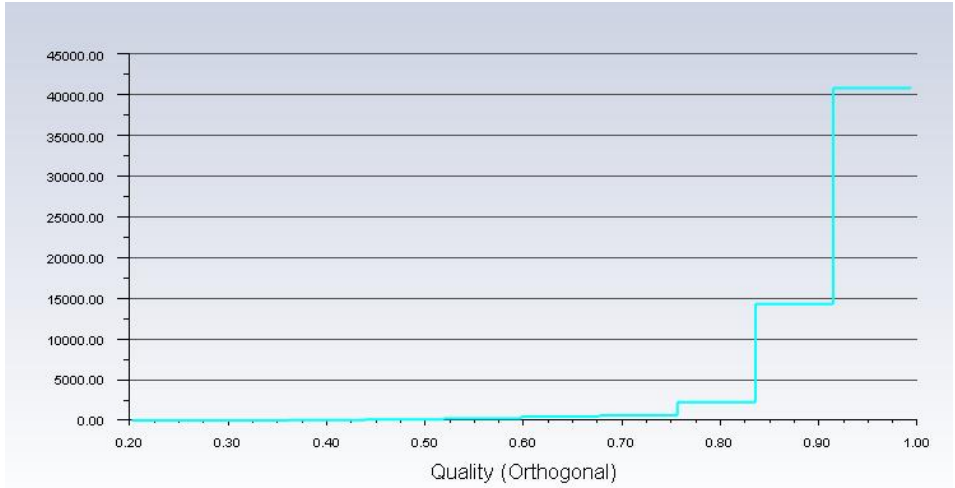


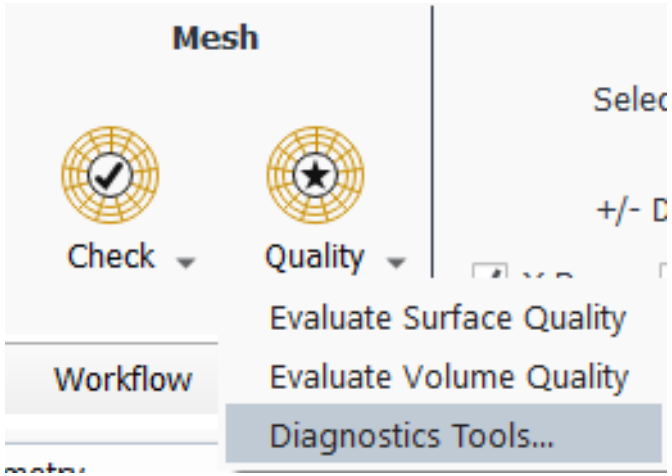
©2022 ANSYS, Inc. / Confidential

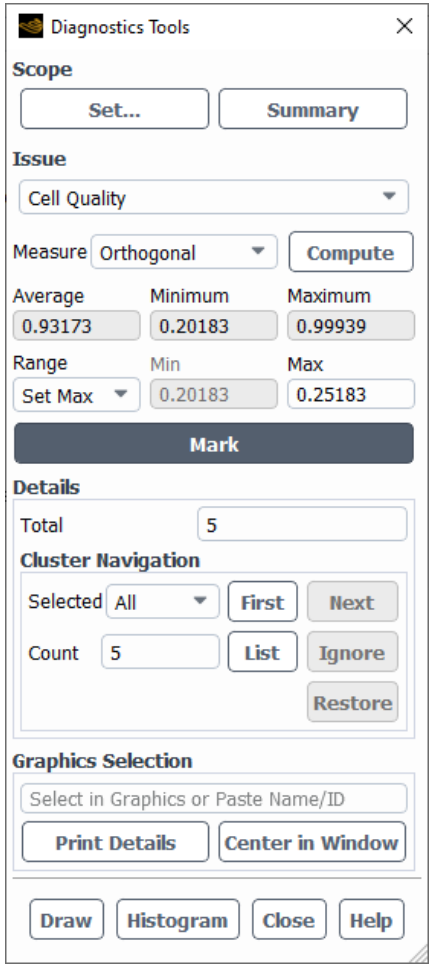
Diagnostics tools

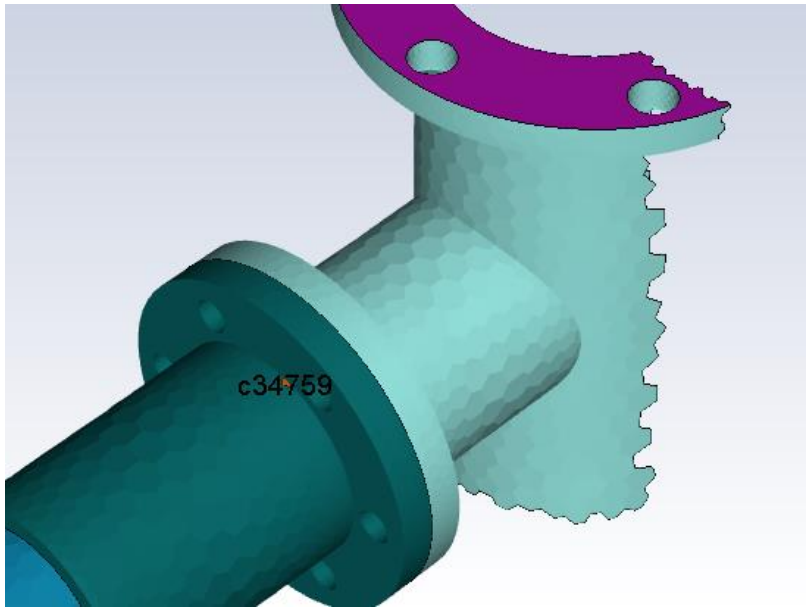
Mesh Diagnostics Tools

- Comprehensive mesh diagnostics tools with visualization
- Surface mesh and Volume mesh characteristics
- Can be used on entire mesh domain or scoped to particular zones/objects







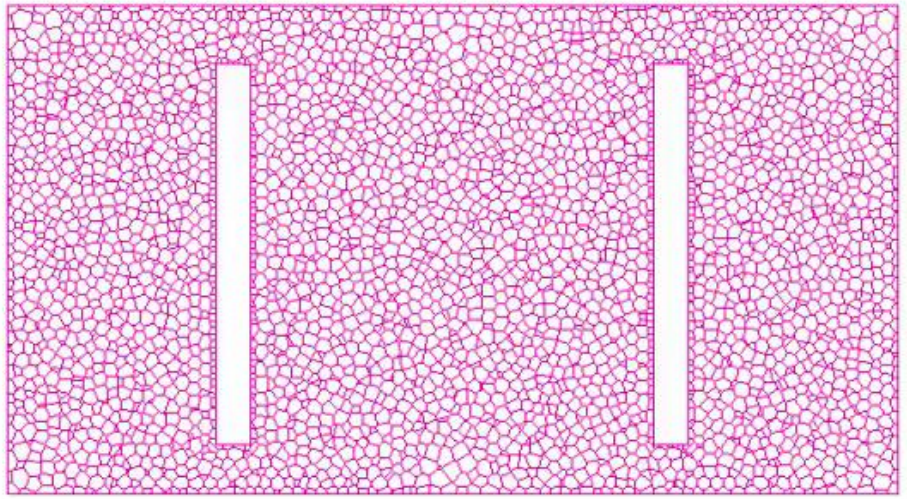


©2022 ANSYS, Inc. / Confidential

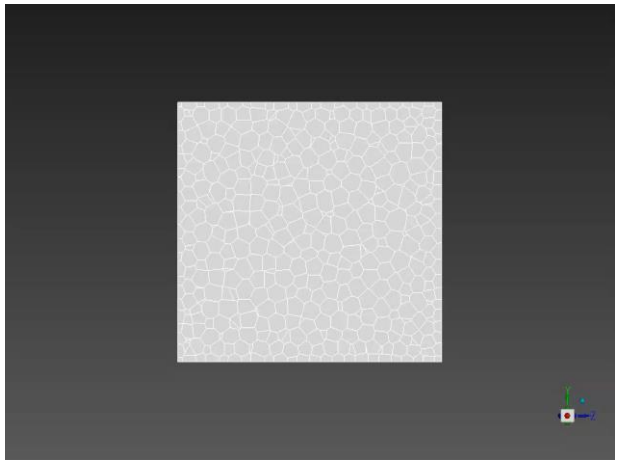
Polyhedral Remeshing

Fluent MDM Poly-remesh

- Good progress made in 2023R1 on the "Poly Remesh" core meshing development and exposure in Fluent
- Cavity remesh
  - with and without boundary remesh
  - with Prism layers
- Best practices being developed
- Work to continue for 2023R2



Mixing blades with poly remeshing



Compressing cylinder with poly remeshing (without smoothing)

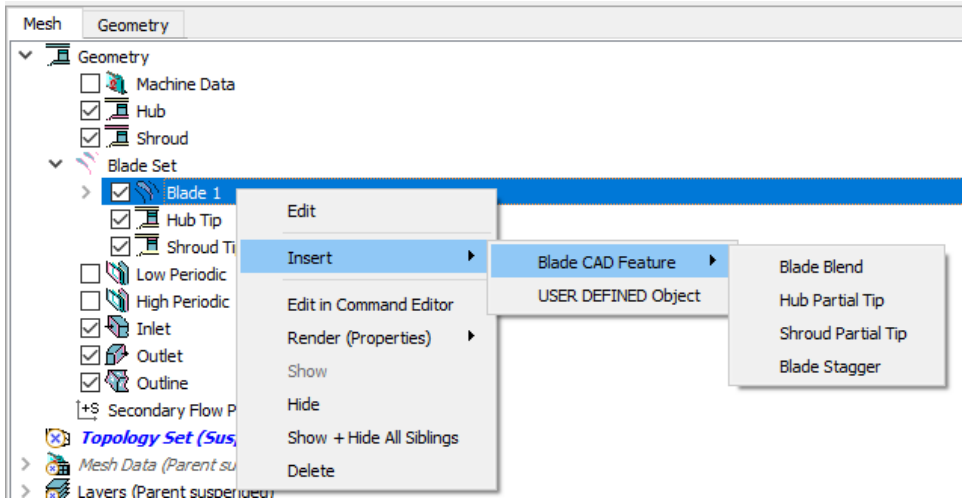
©2022 ANSYS, Inc. / Confidential

TurboGrid

Extending Blade CAD features in TurboGrid

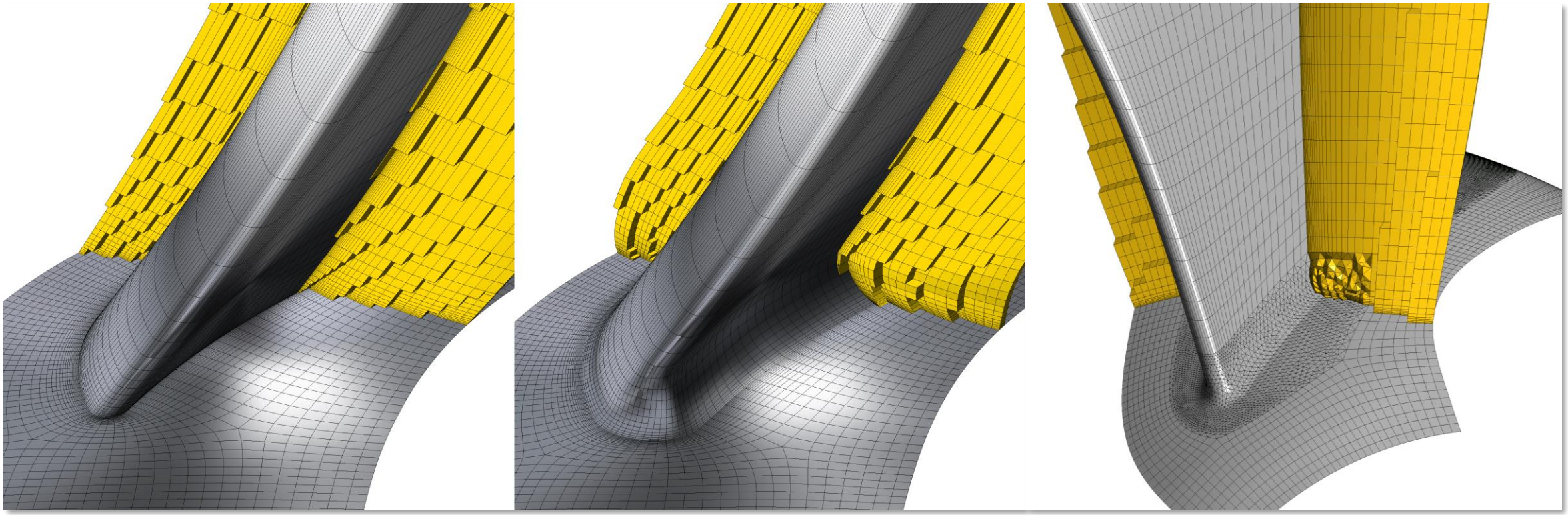
2023 R1 extends the blade CAD features you can create directly in TurboGrid, using only profile/curve files for the geometry input

- Add blade blends (for round LE/TE)
  - Now supports creating true blend geometry when used with hybrid meshing (low-fidelity and high-fidelity geometry created), avoiding the need for an upstream CAD model with blends
  - Also supports approximate blends with low-fidelity geometry for all-hex meshing
  - Approximate or true blends can be defined in a NDF file. Reading the NDF file into TurboGrid will create the blend geometry
- Create partial tip geometry at the hub and shroud
- Adjust the blade stagger angle



©2022 ANSYS, Inc. / Confidential

Adding Blade Blends in TurboGrid



Import profiles/curves and create all-hex mesh

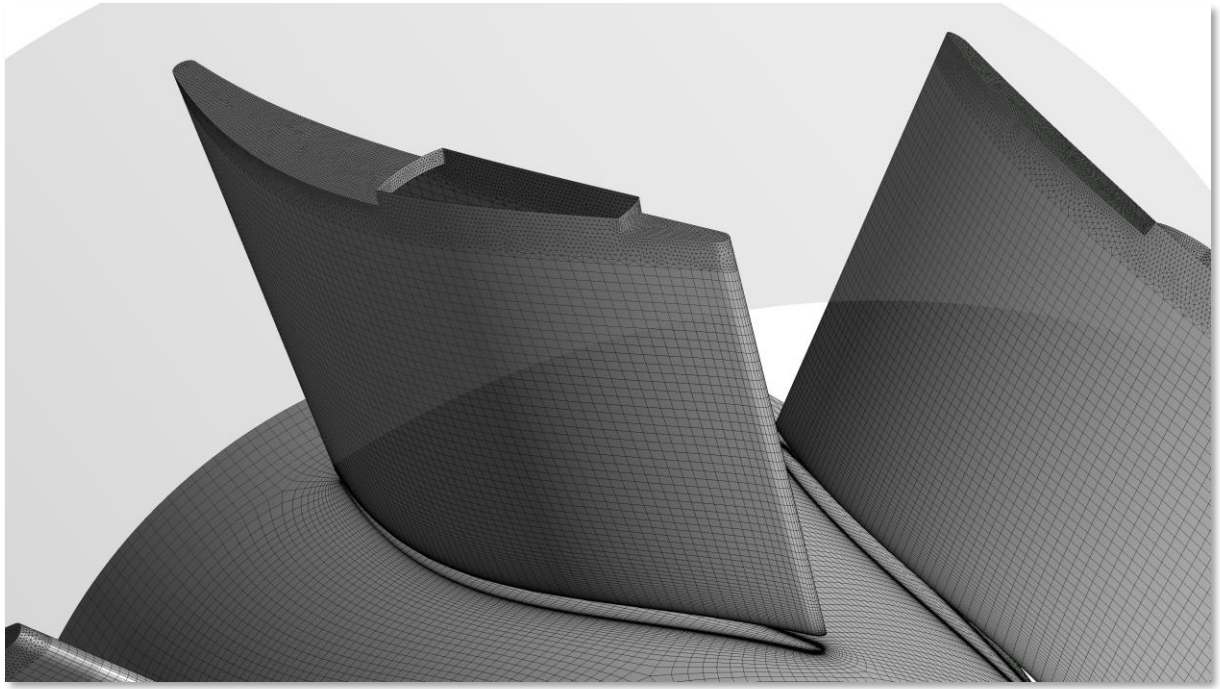
Add approximate blends, retaining all-hex mesh

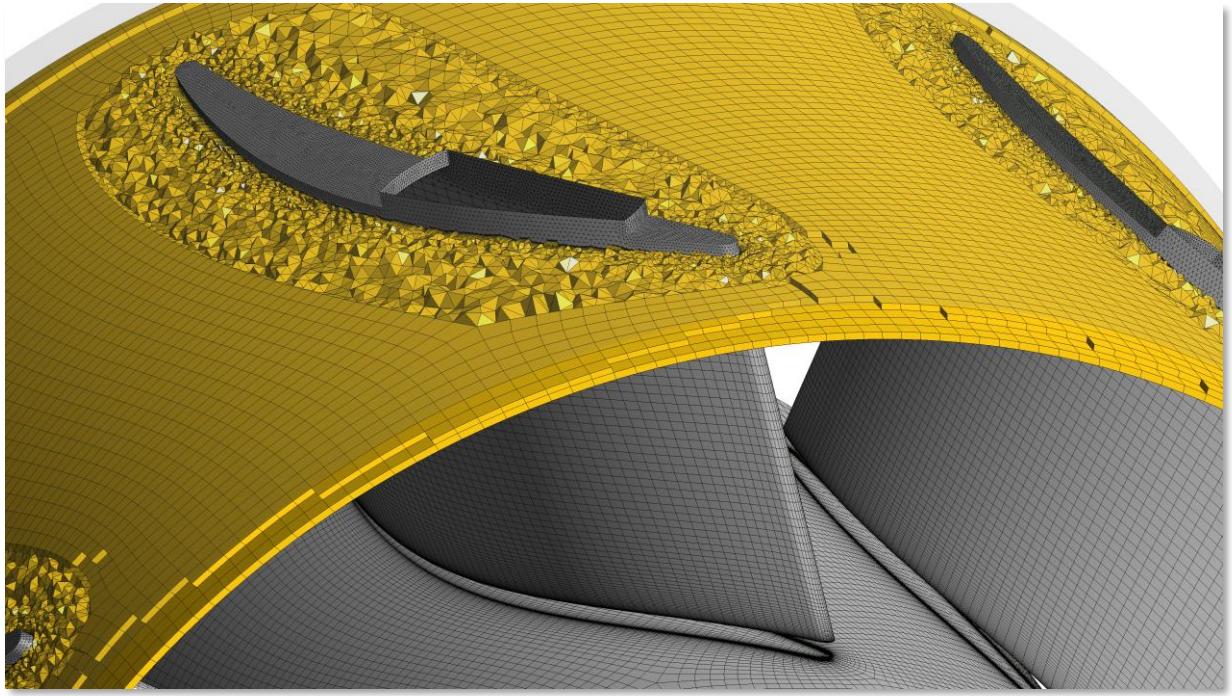
Or TurboGrid can create the high-fidelity CAD representation and add true blends then use fully-automated hybrid meshing

NDF file support for all options

©2022 ANSYS, Inc. / Confidential

Create Partial Tip Gap Geometry in TurboGrid



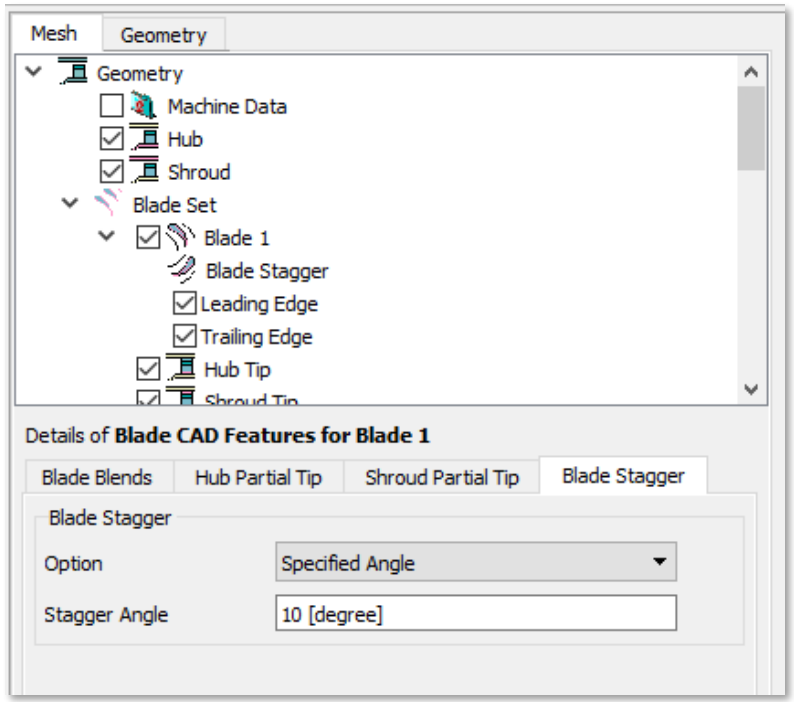


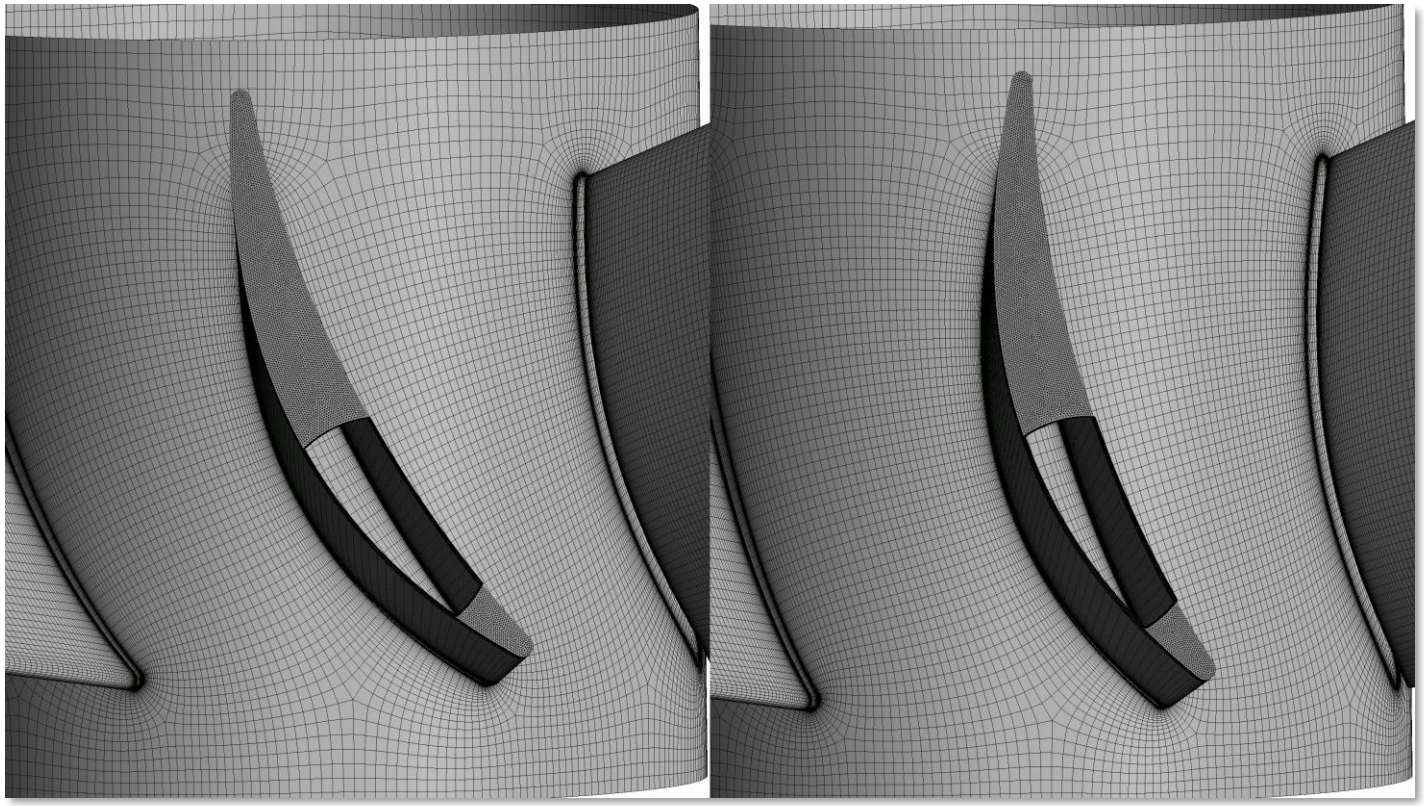
Add the partial tip geometry in TurboGrid at the hub and/or shroud, then automatically mesh with hybrid meshing

©2022 ANSYS, Inc. / Confidential

Adjust Stagger Angles in TurboGrid

Easily adjust the blade stagger angle in TurboGrid without modifying the upstream geometry, then simply update the mesh





Baseline, 0º stagger angle

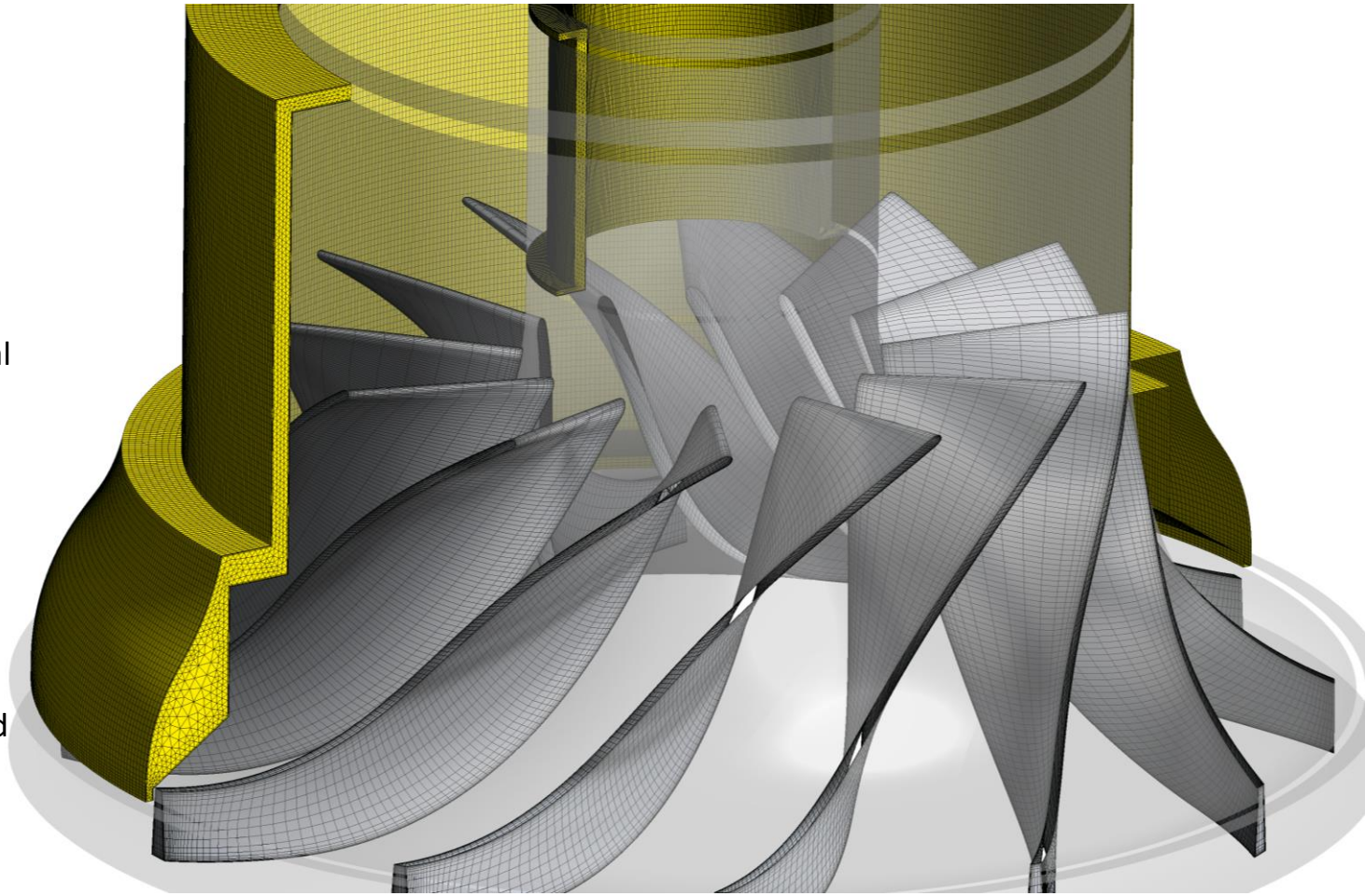
Adjusted, 10º stagger angle

©2022 ANSYS, Inc. / Confidential

NDF: import to TurboGrid & secondary flow path support

NDF (Neutral Data Format) files are an XML definition for blades. The files include the standard blade profiles/curves, plus support for additional features such as blends, and now secondary flow paths

- TurboGrid now supports NDF file import
  - Enables workflows from BladeGen and other blade design tools directly to TurboGrid, without using BladeEditor/Workbench to interactively create additional 3D CAD features
  - All NDF capabilities supported by BladeEditor are supported for NDF import to TurboGrid
  - TurboGrid will automatically create the necessary low-fidelity and/or high-fidelity geometry, depending on the features defined in the NDF file
- Secondary flow path support in NDF files
  - Curves defining secondary flow paths are now supported in NDF files
  - NDF files imported to BladeEditor/TurboGrid will automatically create the secondary flow path geometry/mesh

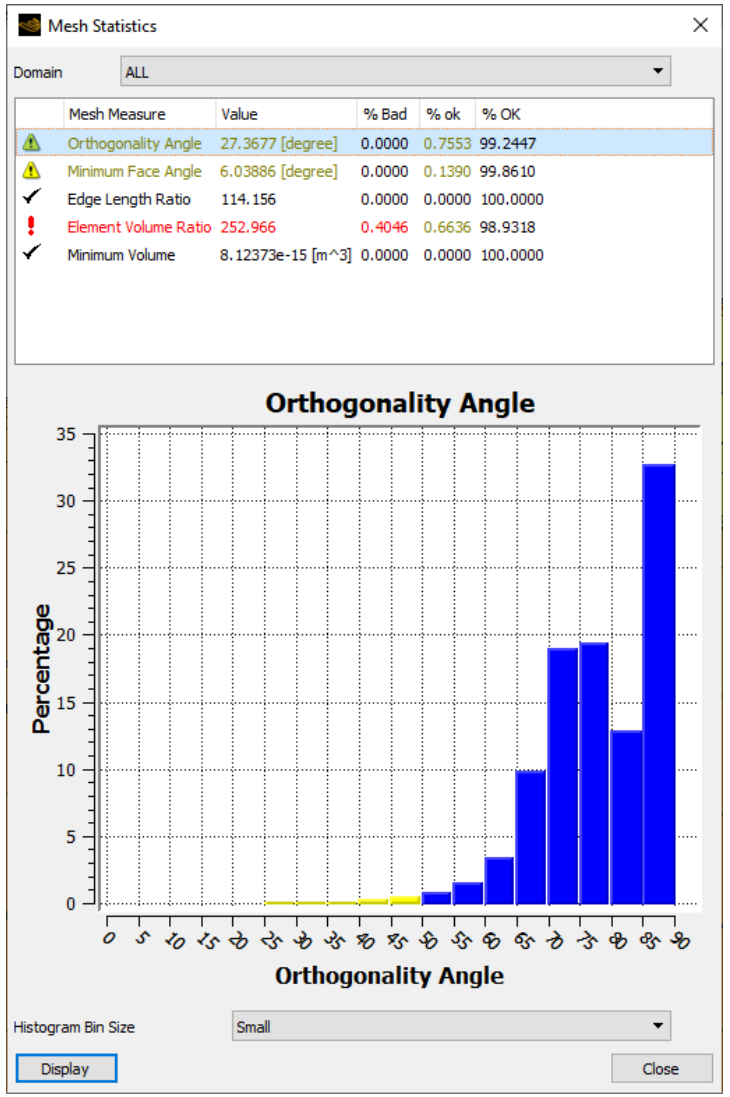


©2022 ANSYS, Inc. / Confidential

TurboGrid Mesh Quality Metrics for Hybrid Meshes

Orthogonality Angle introduced as a new mesh metric

- Orthogonality Angle is the preferred measure for hybrid meshes, instead of Minimum Face Angle
- Orthogonality Angle is calculated in the same way as the CFX Solver, to ensure consistency
- Mesh statistics that are not of interest can now be hidden



©2022 ANSYS, Inc. / Confidential

SpaceClaim Interactive Meshing

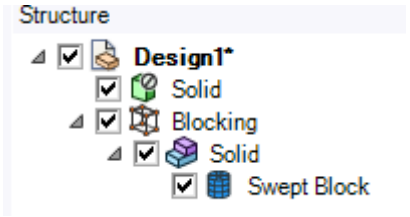
SpaceClaim Meshing – New Features

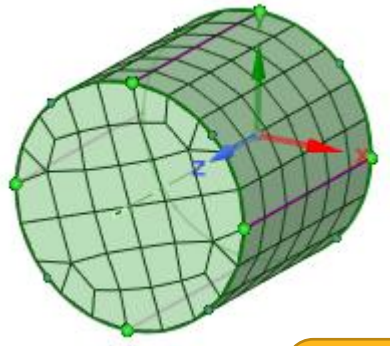
- Usability improvements
- Performance/Robustness/Quality improvements
- Support for automatic MultiZone meshing of
  - Axis symmetric bodies
  - Torus

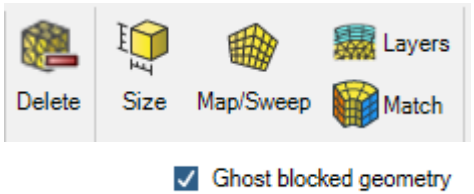
©2022 ANSYS, Inc. / Confidential

SCIM: Smart selection options for blocking

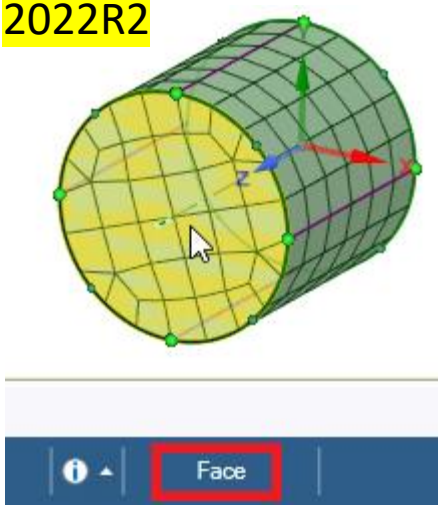
- Common user complaint: Selection of geometry Vs blocking is confusing/difficult

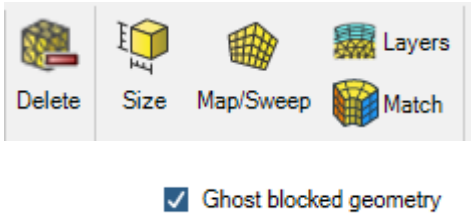




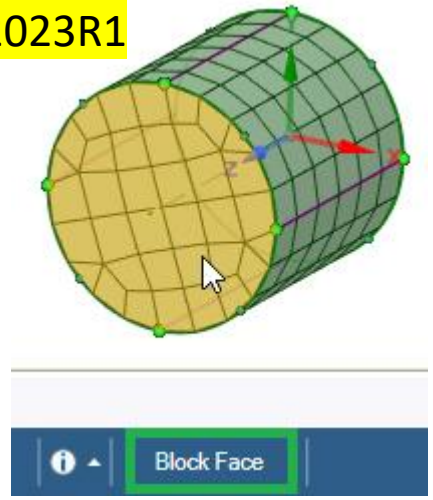


2022R2





2023R1



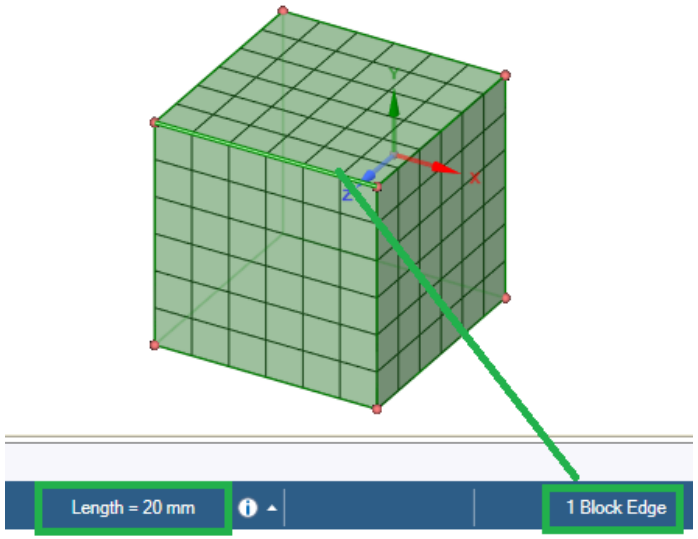
In 23R1 only blocking entities are selected when **Ghost blocked geometry** option is enabled

©2022 ANSYS, Inc. / Confidential

SCIM: Block Information Usability

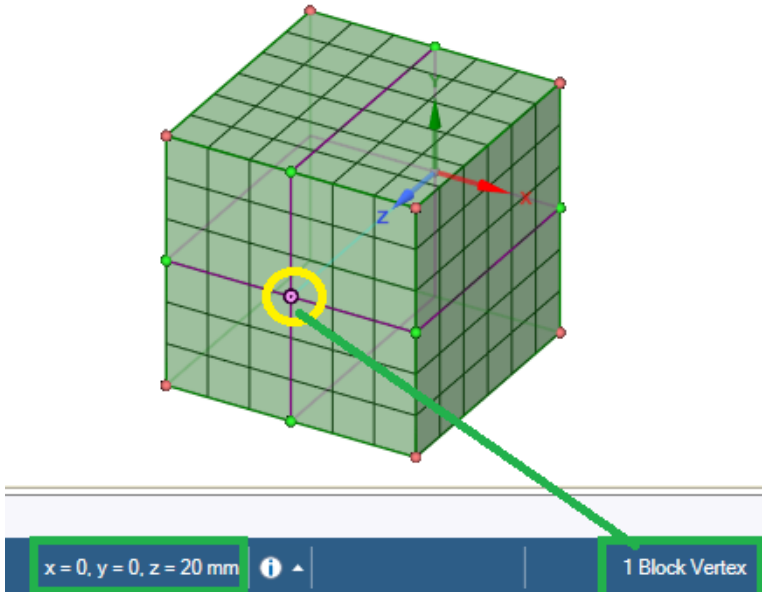
When selecting a block edge

Show length of block edge



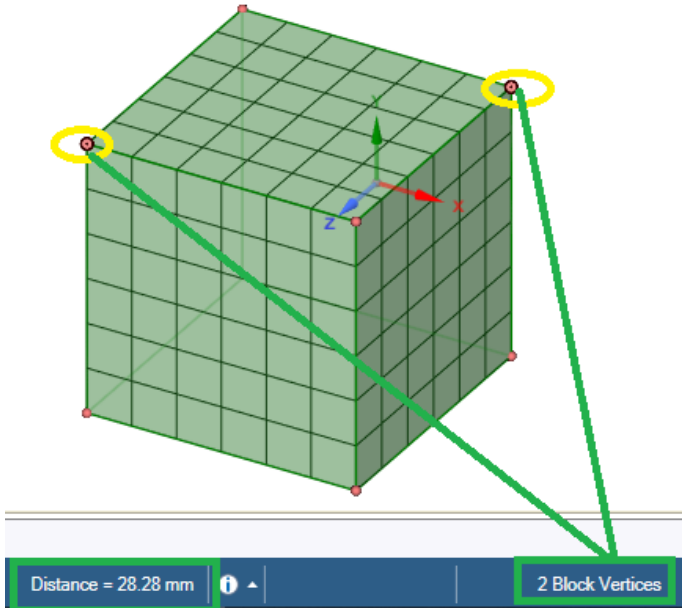
When selection a block vertex

Show position of block vertex



When selection 2 block vertices

show distance of the block vertices

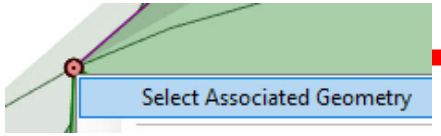


Show measurement information for block entities in status bar

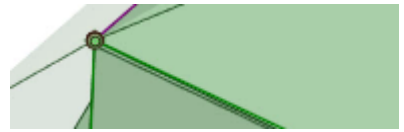
©2022 ANSYS, Inc. / Confidential

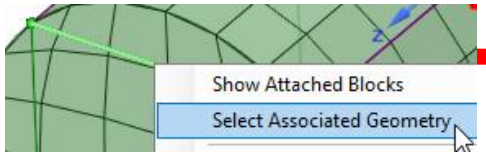
RMB → Show Associated Geometry

This function works for all blocking entities

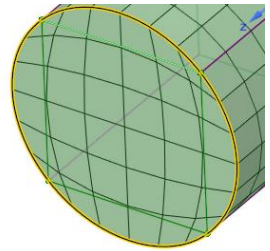


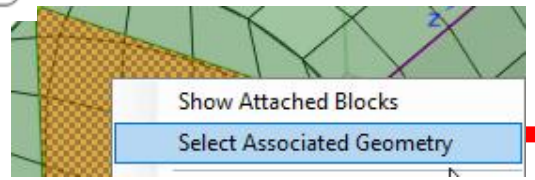
BlockVertex =>Select Associated Geometry=> **Vertex**



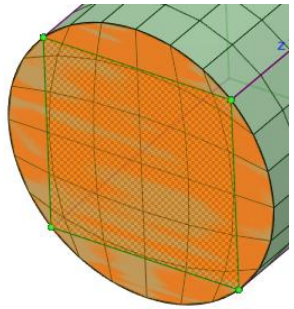


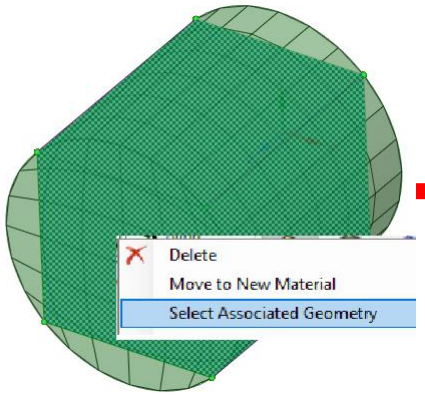
BlockEdge =>Select Associated Geometry=> **Edge**



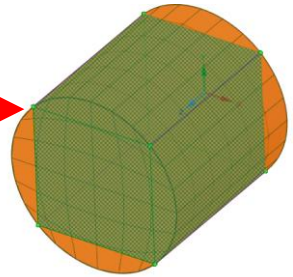


BlockFace =>Select Associated Geometry=> **Face**





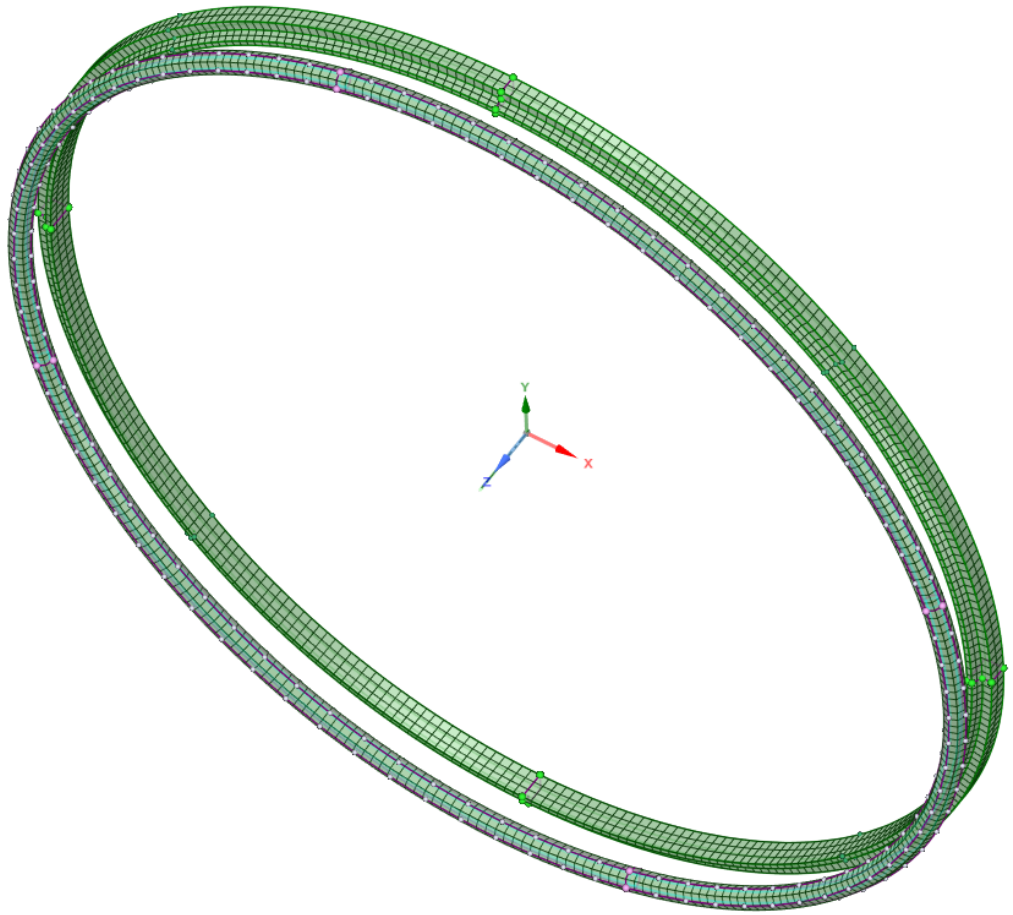
Block =>Select Associated Geometry=> **Body**



©2022 ANSYS, Inc. / Confidential

SCIM: General Improvements

- Wide range of robustness improvements
- Performance improvements for large models
- Improved default blocking results
- Good, automatic meshing of axis-symmetric bodies and tori



©2022 ANSYS, Inc. / Confidential

PyPrimeMesh

PyPrimeMesh (Beta) – New Features

- Surface meshing
  - Constant size quad
  - Variable/constant size triangles
  - Size field
  - Quality based localized remeshing
  - Threaded parallel
- Wrapper
  - Fault tolerant meshing
- Volume meshing
  - Tet, Poly, Hexcore, Poly Hexcore, Prism
    - Size field
    - Auto node move
    - Distributed parallel
- Connections
  - Join, intersect, stitch (facet based)
  - Scaffolding (topology based)
  - Subtract (facet based, later topology)
- Misc
  - Labels, Zones (collections, settings)
  - Parts (model organization)
  - Transforms
- Import/Export
  - CAD
  - Fluent: .cas, .msh
    - Size field
  - CDB
- Graphics
  - PyVista

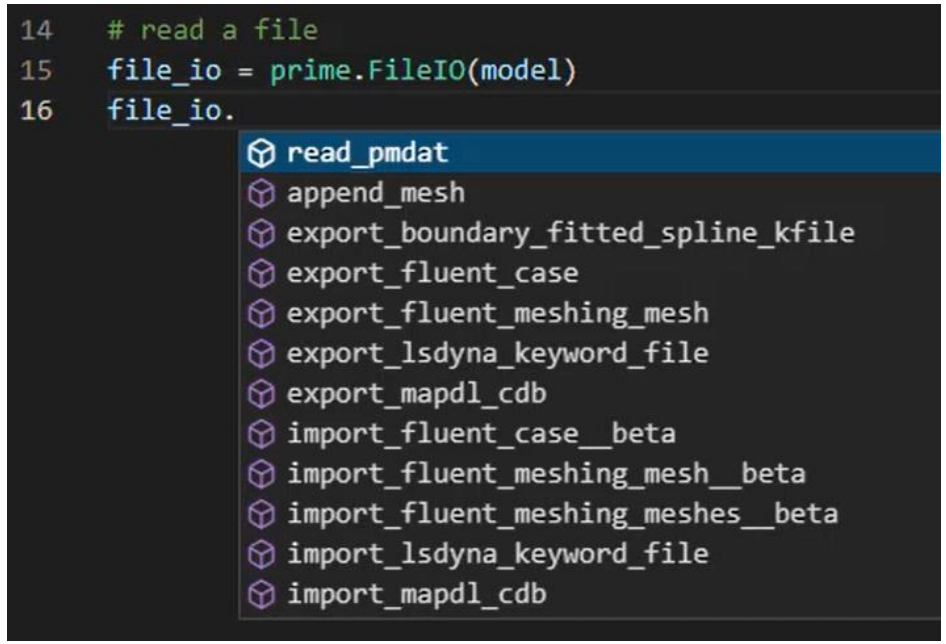
©2022 ANSYS, Inc. / Confidential

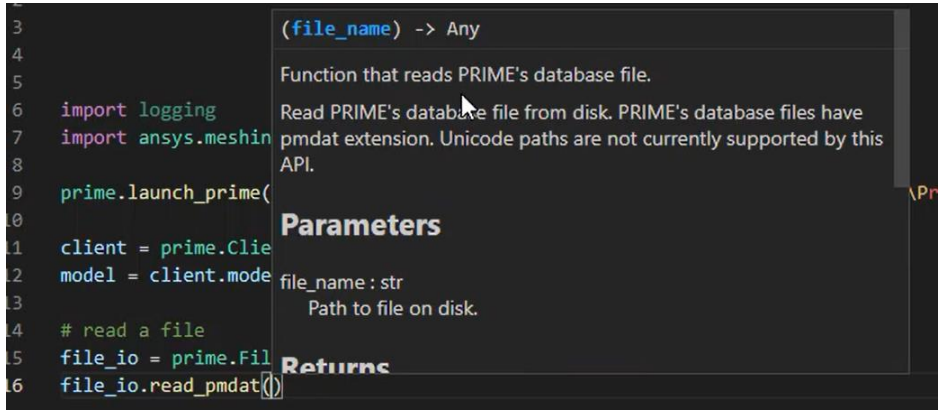
PyPrimeMesh & Ansys Prime Server

PyPrimeMesh lets you use Ansys core meshing technology, embedded across Ansys flagship tools, directly from within your python environment.

- PyPrimeMesh (Open Source Client on Github)
  - Autogenerated code + low level API's (~160 API's)
  - High level API's (Lucid)
  - Graphics (PyVista)
  - Examples
- Ansys Prime Server (Server provided in Unified Ansys install)

- In 2023R1 with PyPrimeMesh BETA, it's possible to:
  - Generate surface and volume mesh of various types using parallel capabilities
  - Apply a range of sizing controls to control mesh distribution
  - Create complex workflows and automation
  - Use wrapping methods to extract and mesh regions
  - Modify connectivity of topology and mesh
  - Import and facet CAD
  - Export to various Ansys solvers



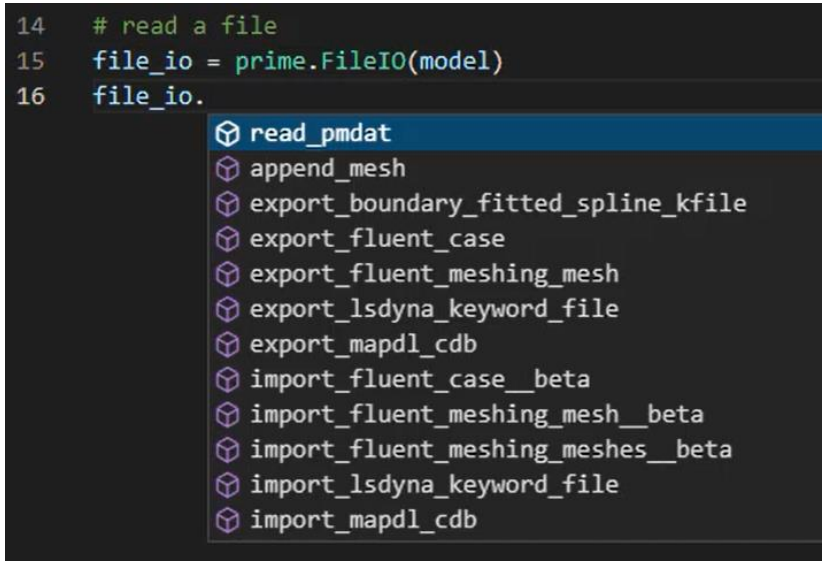


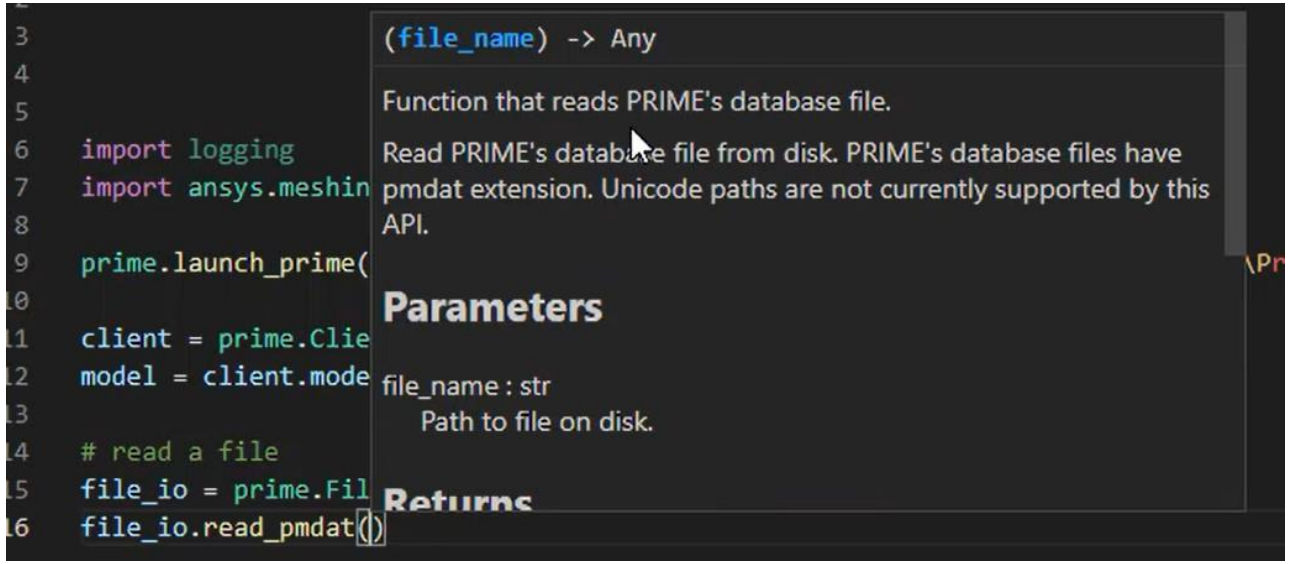
©2022 ANSYS, Inc. / Confidential

2023R1 PyPrimeMesh @ beta – Main Feature Set

- Clean geometry
- Dirty geometry: Wrapper
  - Use wrapper for cleanup

- Usability features
  - IDE support needed
  - Auto completion
  - Documentation available at the time of writing scripts
  - Hidden features
  - …
- Release
  - License
    - Server checks cfd_preppost, preppost, (or advanced_meshing)
  - GitHub
    - Client code
    - Manual
    - Tutorials



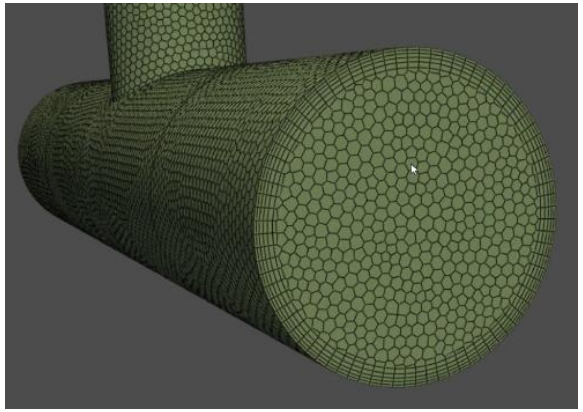


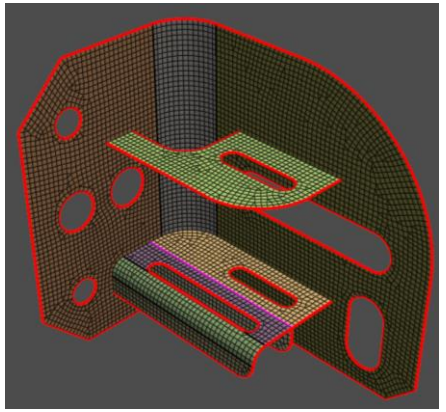
©2022 ANSYS, Inc. / Confidential

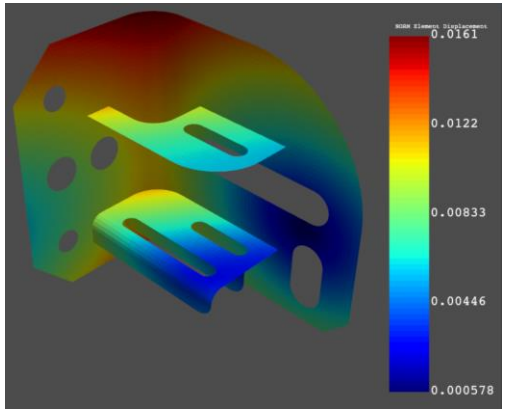
PyPrimeMesh Examples

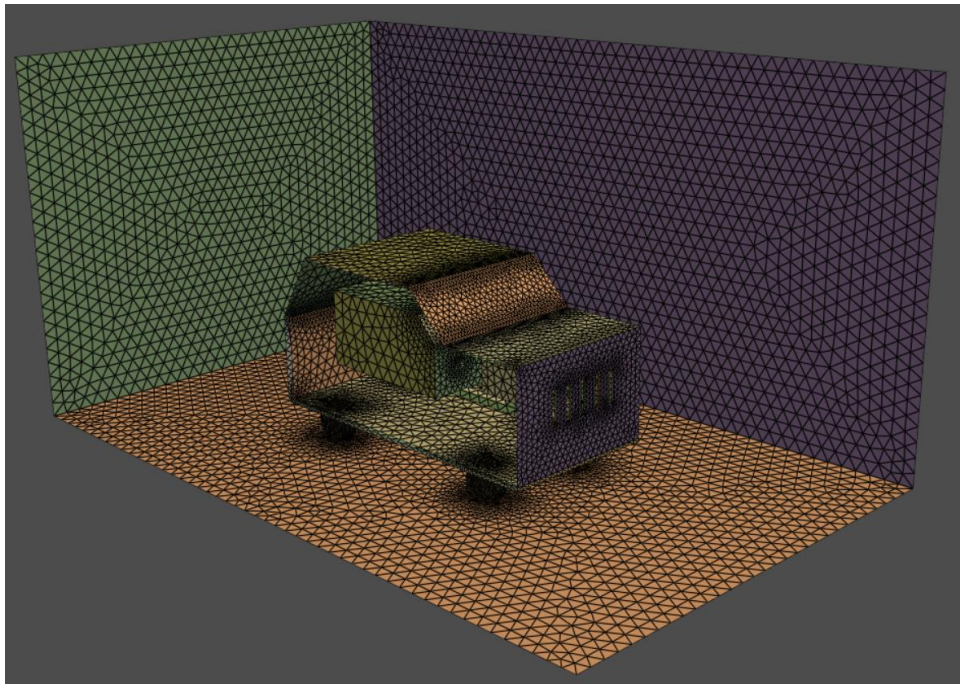
Some examples:

- Wrapping
- Poly mesh with inflation
- Quad shells
- Connecting topology
- …





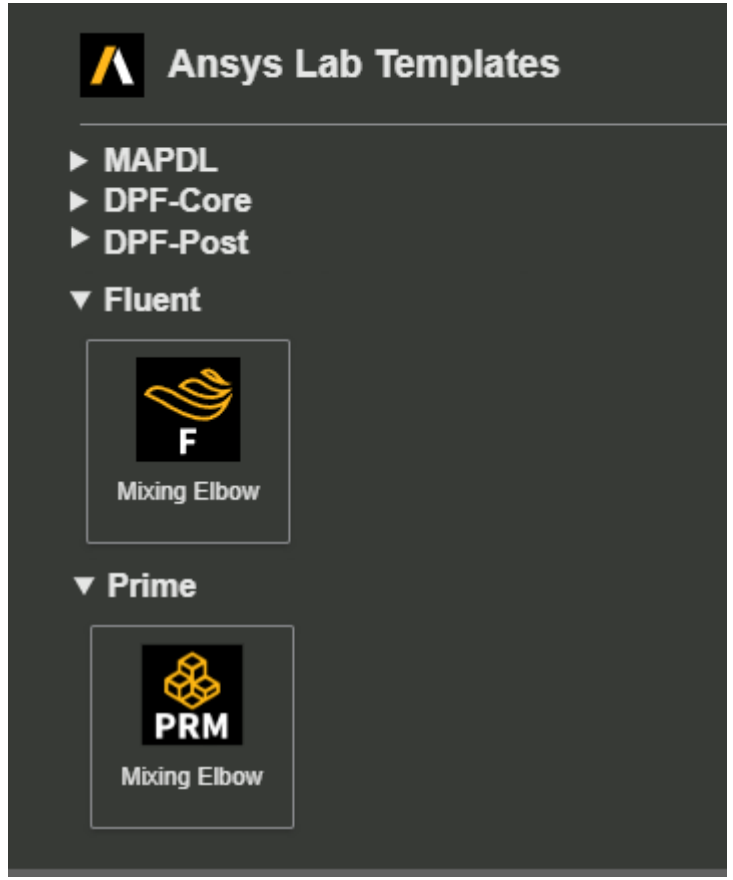




©2022 ANSYS, Inc. / Confidential

PyPrimeMesh on Ansys Labs

- PyGeometry will soon be able to export .fmd format to provide a route for CAD import
- Native CAD import options can also be used
  - STL, ACIS, Parasolid
- PyVista graphics buttons not currently supported in Jupyter



©2022 ANSYS, Inc. / Confidential

Ansys