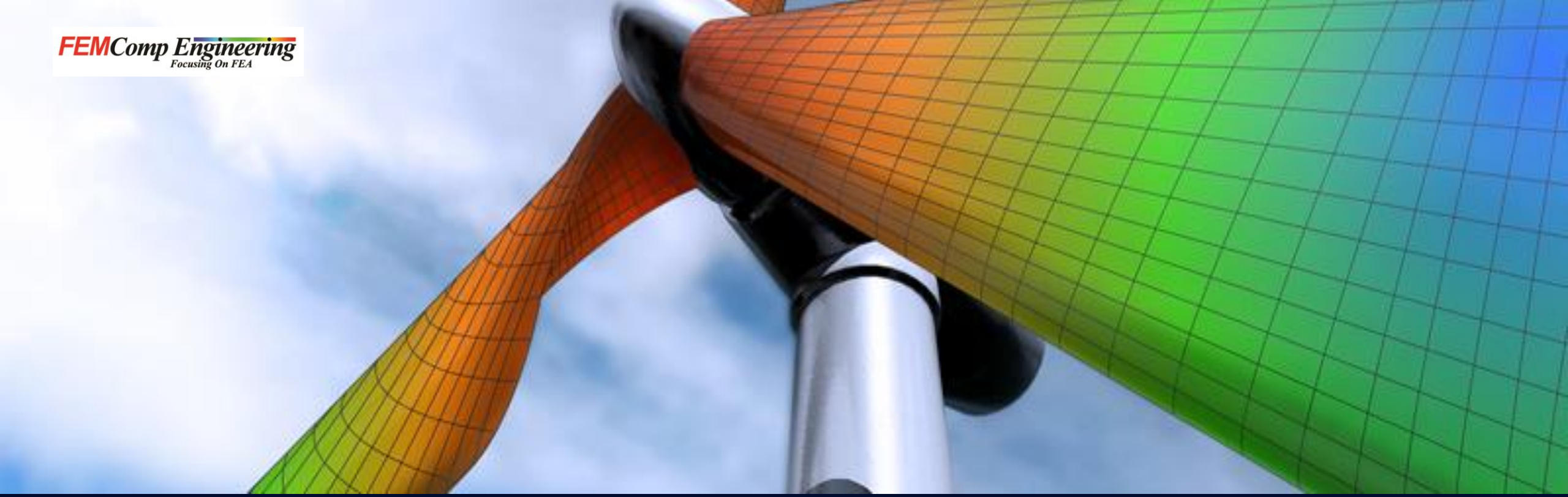
FEMComp Engineering
Focusing On FEA



Simcenter Femap v.2021.2
Introduction to **Body Mesher Technology**
Webinar

Presenter: **Tadeusz Chmielewski**
CEO of FEMComp Engineering AB

SIEMENS

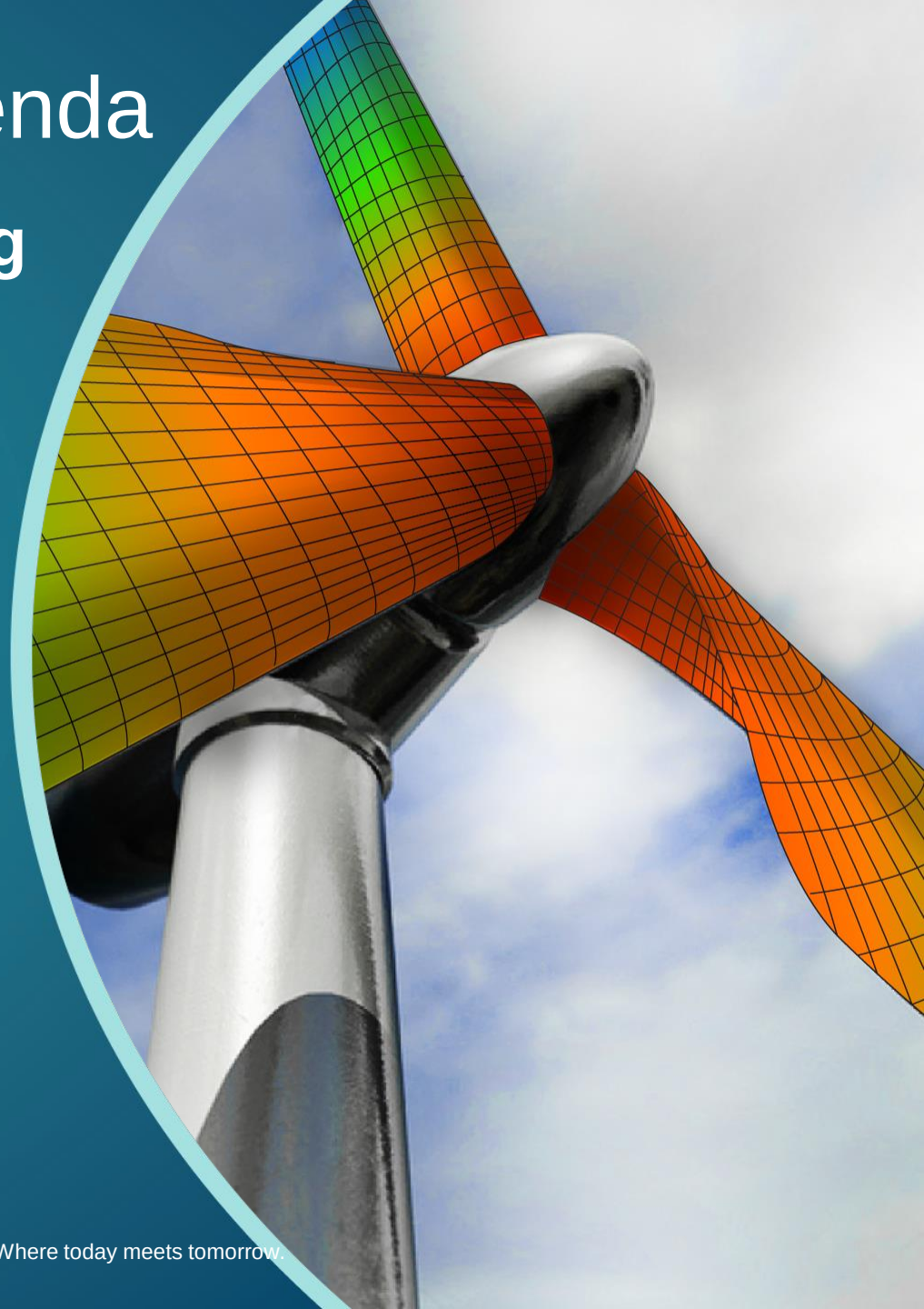
Body Mesher Technology - Agenda

Faceted Geometry/Convergent Modeling

- Overview
- Import/Export and Identification
- Facet Body Operations

Body Mesher Technology

- Why Body Mesher?
- Traditional vs. Body Meshing Workflow
- Mesh, Bodies command
- Body Meshing Suggested Workflow
- Body Meshing Benefits
- Powerful Transition Meshing
- Mesh, Mesh on Mesh command



Faceted Geometry/Convergent Modeling
Overview

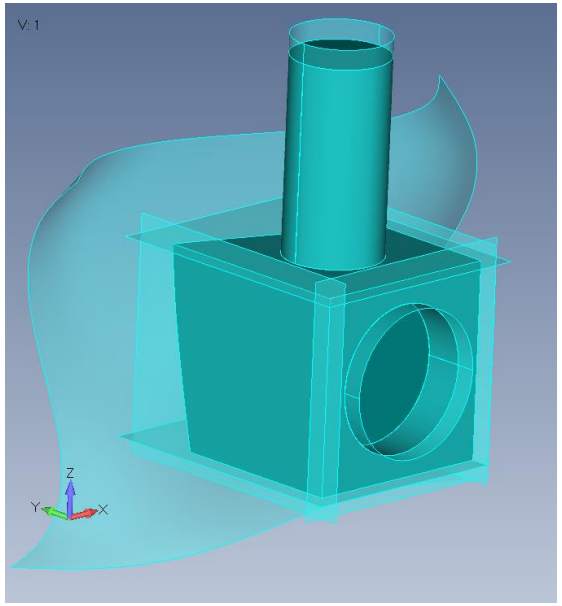
Femap v2021.2 adds support for Parasolid **Convergent Modeling**

- **Convergent Modeling** works with facets, surfaces, and solids without any conversion
- This means you can import **STL**-file or **scanned** data to Femap as a **convergent body** then use traditional modeling commands
- You can then **create geometry** on or around the facet
- Faceted or Convergent Modeling **extends** Parasolid, and hence Femap's surface and solid modeling to include models where the underlying geometry is comprised of **triangular facets**

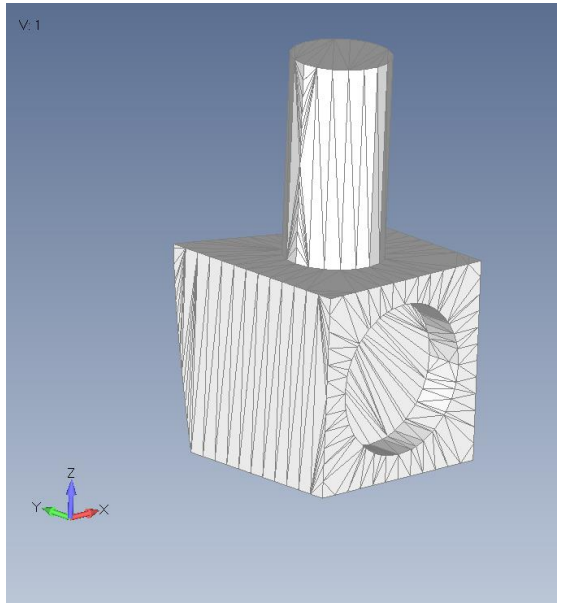
Faceted Geometry/Convergent Modeling
Overview

Faceted or **Convergent** Modeling **extends** Parasolid, and hence Femap's surface and solid modeling to include models where the underlying geometry is comprised of **triangular facets**

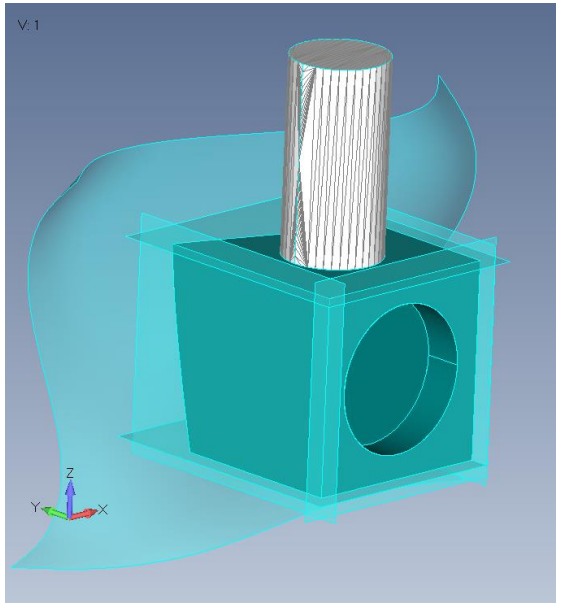
Note: Femap doesn't support the **creation** of convergent models, it **does** support **import/export**, **editing** and **meshing**



Traditional Parasolid B-REP, faces of the solid are defined by underlying analytic and/or NURBS surfaces



In Convergent Modeling, the underlying surfaces are made up of triangular facets



Convergent Models can contain a mixture of traditional and facetted faces

Faceted Geometry/Convergent Modeling
Import/Export and Identification

FEMComp Engineering
Focusing On FEA

Import/Export

Text-Based Parasolid Files

.x_t – contains topology

.m_t – contains the facets

Binary

.x_b – file includes both

Identification

Model Info Tree

Listings

Type: Solid

Type: Solid Partial Mesh

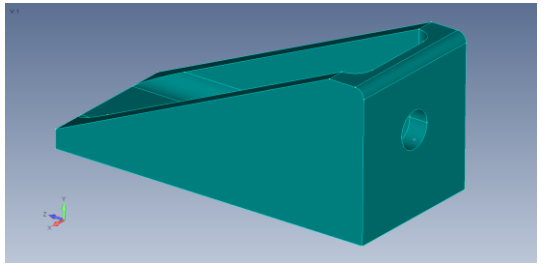
Type: Solid Full Mesh

SIEMENS

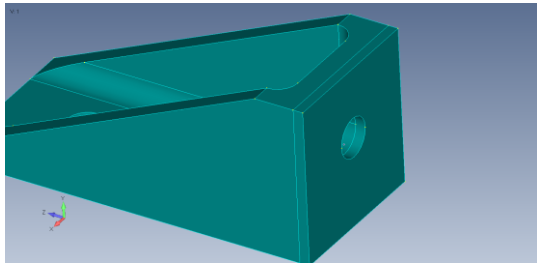
Faceted Geometry/Convergent Modeling
Facet Body Operations

FEMComp Engineering
Focusing On FEA

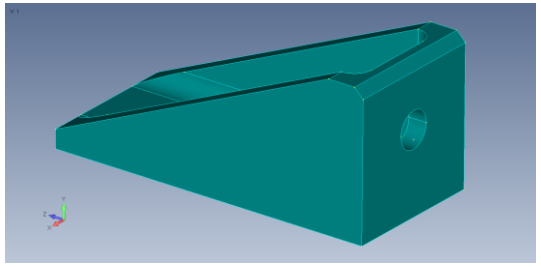
Blend/Fillet



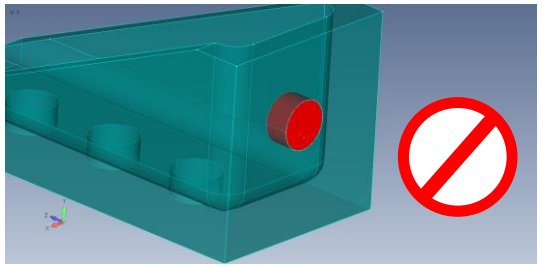
Thicken



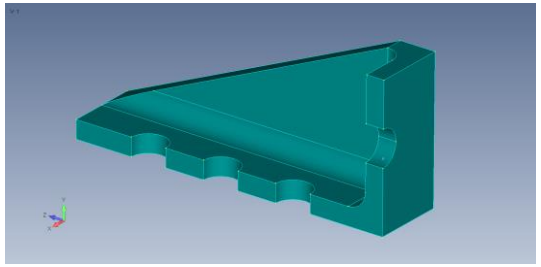
Chamfer



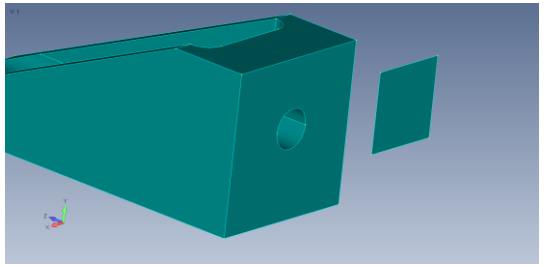
Fill Hole



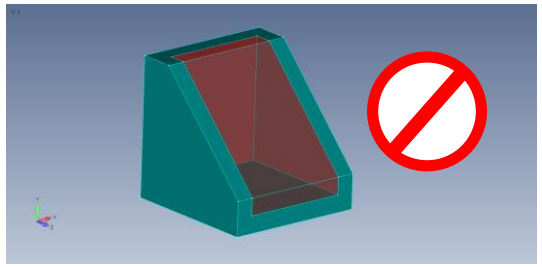
Slice



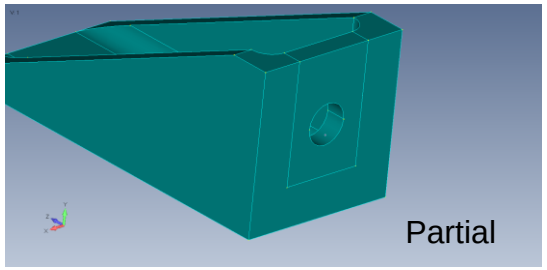
Extend



Hollow/Shell



Embed Face

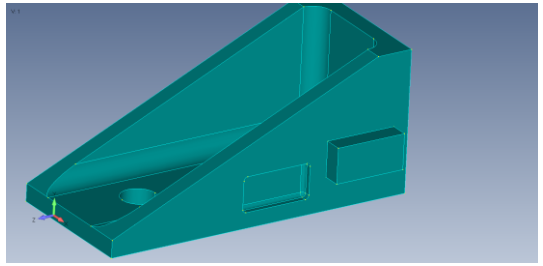


SIEMENS

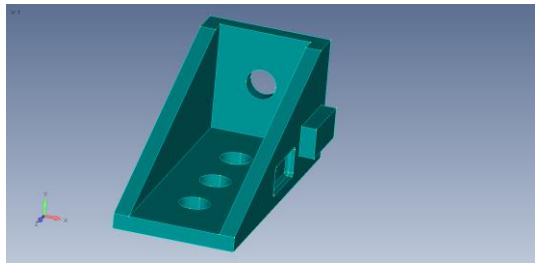
Faceted Geometry/Convergent Modeling
Facet Body Operations

FEMComp Engineering
Focusing On FEA

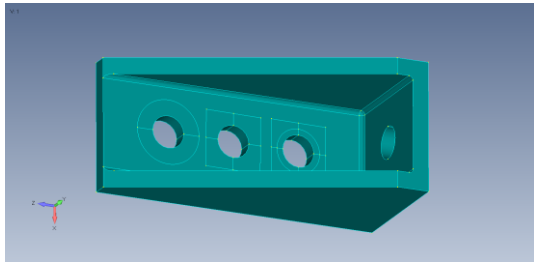
Boolean Add/Subtract



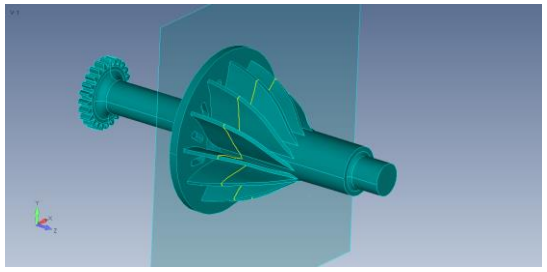
Remove Face



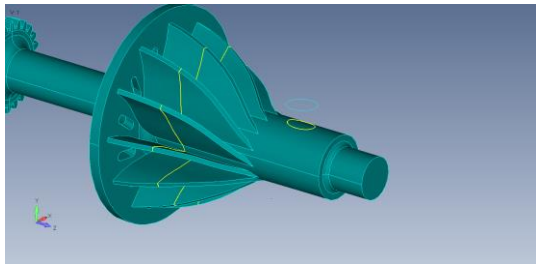
Pad and Washer



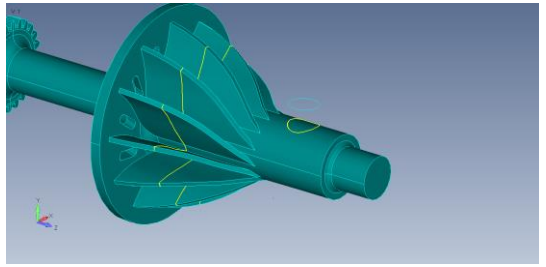
Curve From Surface Intersect



Curve From Surface Project Normal



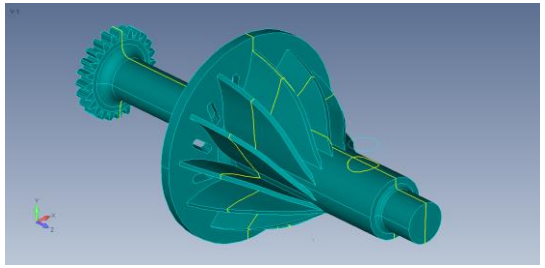
Curve From Surface Project Along Vector



Curve From Surface UV Curves

No UV Space on Facet Faces

Curve From Surface Slice



SIEMENS

Faceted Geometry/Convergent Modeling
Surface Meshing Example

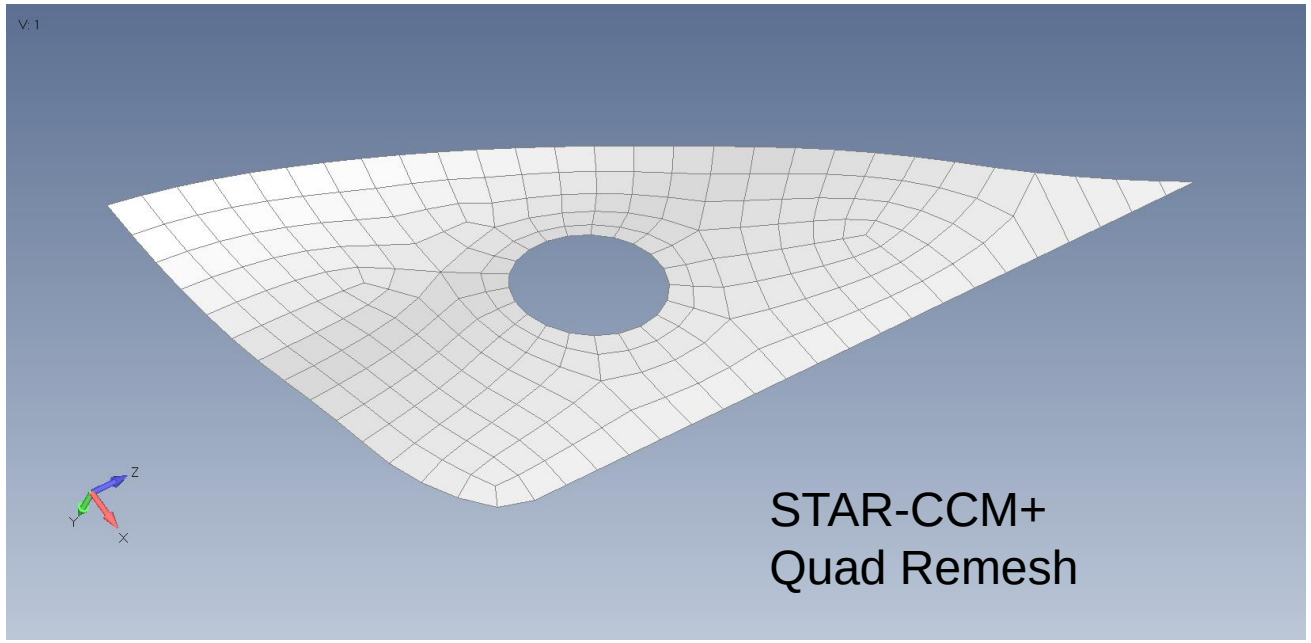
FEMComp Engineering
Focusing On FEA

Challenge – Surface Mesh complex Additive Manufacturing design models with a quality Mesh

Answer – Triangle “Remeshing” technology from **STAR-CCM+**

The STAR-CCM+ **Triangle** Remesher **integrated** into FEMAP turns poorly shaped triangles into high quality triangles

During implementation, we also realized how well it can create a Quadrilateral-dominant mesh, thus the result is Femap’s new “**Body Mesher Technology**”



SIEMENS

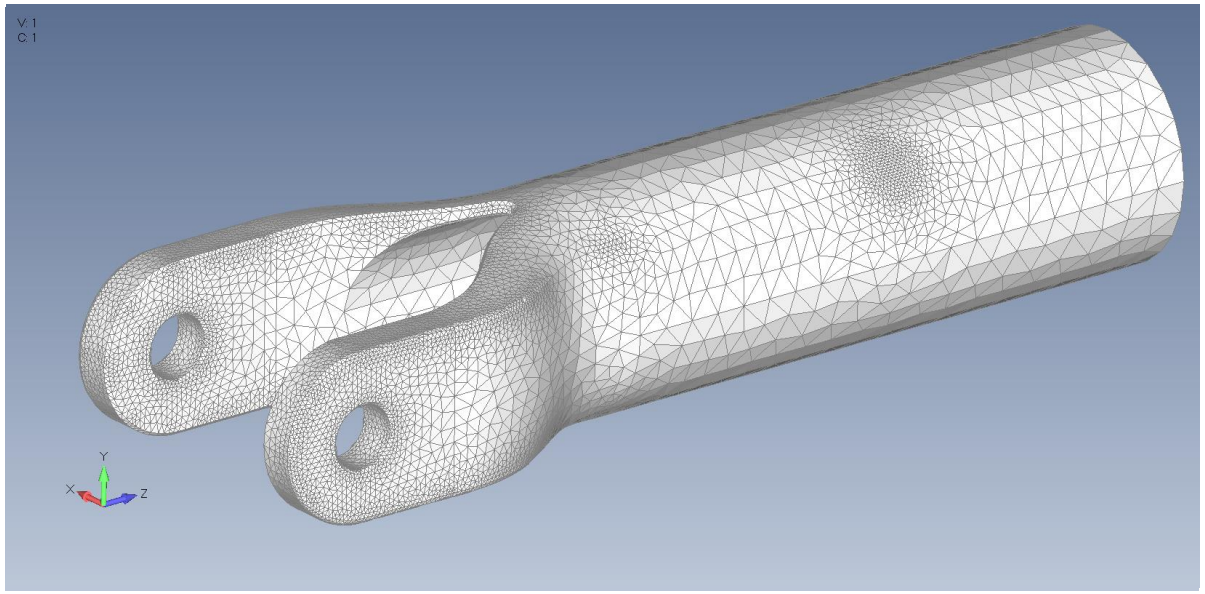
Body Mesher Technology
Why Body Mesher?

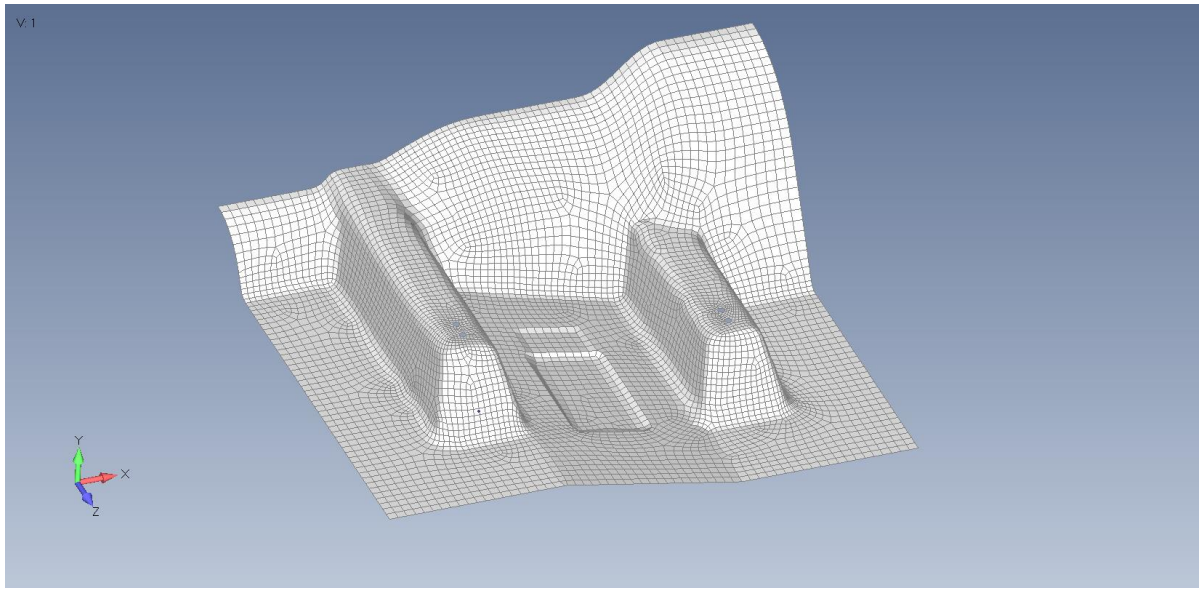
Why did we decide to name this functionality the “**Body Mesher**”?

Because you mesh a Parasolid Body, which can either be a connected set of surfaces that form a single closed volume (**solid** body) or that do not a **sheet** body or **general** body

Commands which always use or optionally allow use of **Body Mesher** Technology:

- **Mesh, Bodies** (always)
- **Mesh, Mesh on Mesh** (always)
- **Mesh, Geometry, Surface** (optional)





Body Mesher Technology
Traditional Meshing vs Body Meshing Workflow - Overview

FEMComp Engineering
Focusing On FEA

Body Meshing can help to create very **high quality** meshes
The workflow is very **different** from traditional meshing in FEMAP

Traditional Meshing Workflow

- **Idealize geometry** via Meshing Toolbox and/or Geometry Preparation to remove slivers and geometry problems
- Define **discrete**, highly customizable **mesh locations** on geometry

- **Nodal locations** are primary, guaranteed result, therefore the best **quality** mesh is sometimes **compromised**

Body Meshing Workflow

- **Not Required**, Body Mesher handles this via triangulation
- Define **Target Mesh Sizing**, along with Element **Quality** Targets and Tolerances

- **Mesh quality** is primary goal
- Node locations are therefore **not** guaranteed, as any discrete mesh sizes are targets

SIEMENS

Body Mesher Technology
Traditional Meshing vs Body Meshing Workflow - Associativity

Because Nodes may or may not end up on curves and surfaces, **geometry associativity** is **affected**

Traditional Meshing Workflow

- Nodes and Elements are **fully associated** with Points, Curves and Surfaces of Surface and Solid Meshes

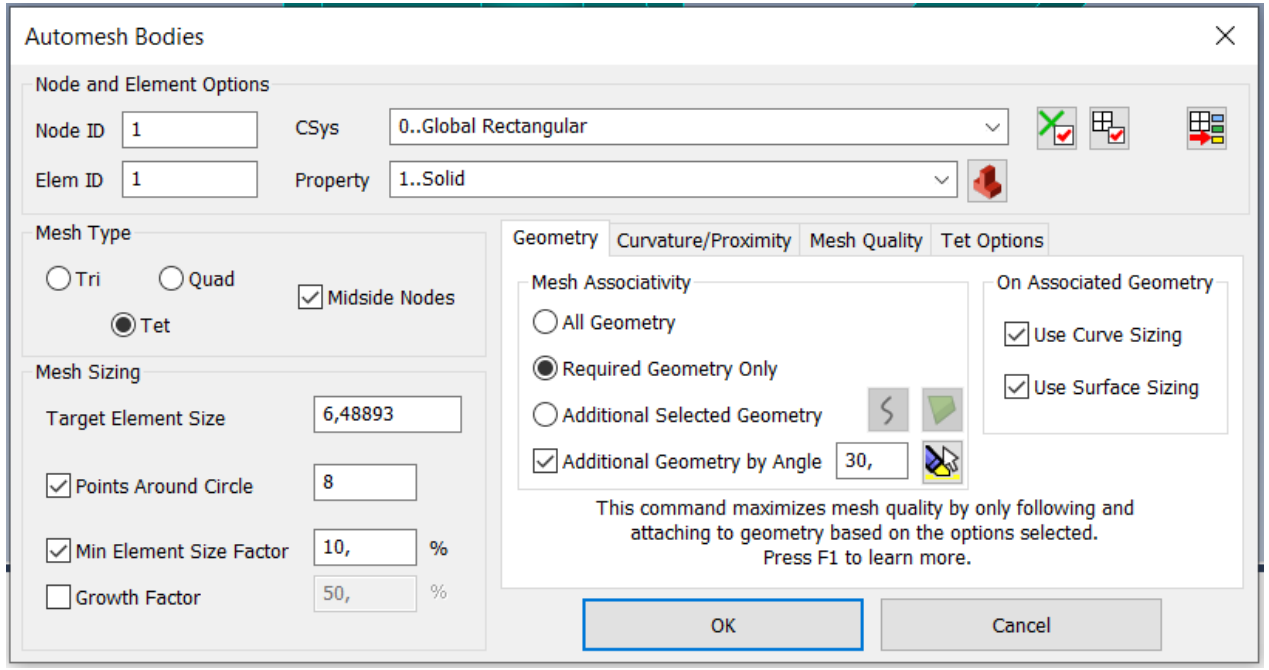
Body Meshing Workflow

- Nodes and Elements are **only associated** with Points, Curves, and Surface where there are **existing Loads**, **Boundary Conditions**, or **Regions** (including **contact**)
- Nodes and Elements **can** also be associated with additional Curves and Surfaces **explicitly** selected by **user**

Body Mesher Technology
Mesh Bodies…

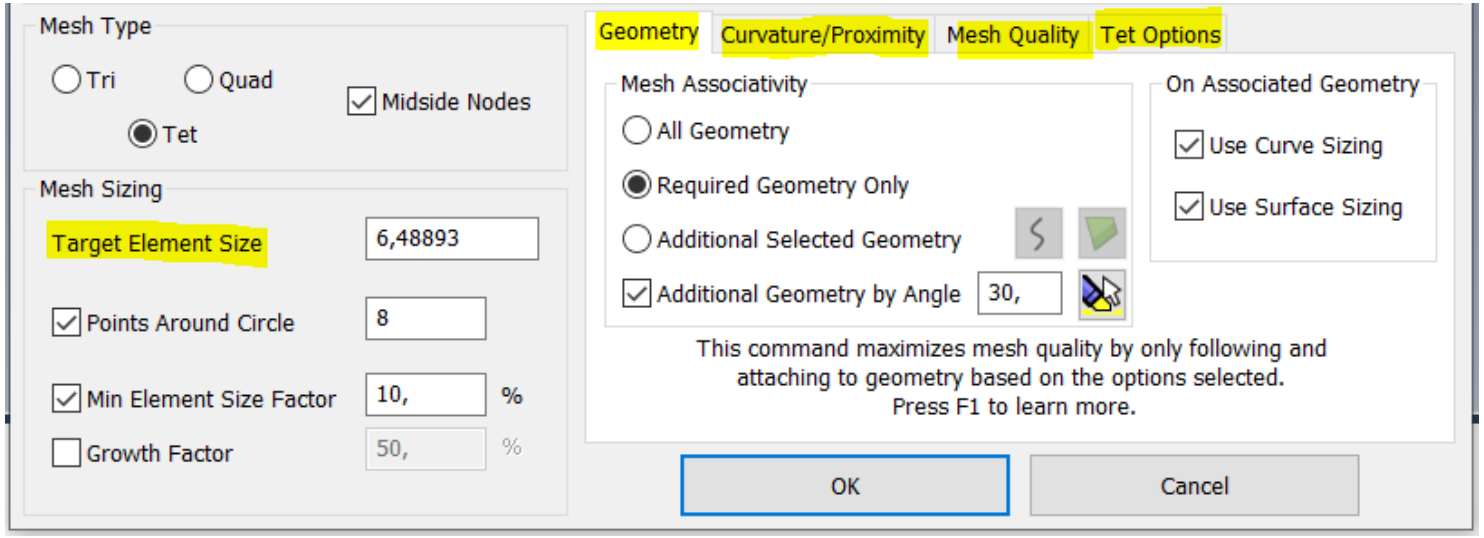
- Uses meshing methodologies which **differ** from Mesh, Geometry, Surface and Mesh, Geometry, Solid commands for creation of a higher quality mesh by imposing **fewer restrictions** when meshing geometric surfaces (sheet solids), solids, and general bodies

- Can create a **2-D surface mesh** on a connected sheet solid (stitched body) or general body or a **3-D solid tetrahedral mesh** in a solid part



Body Mesher Technology
Mesh Bodies…

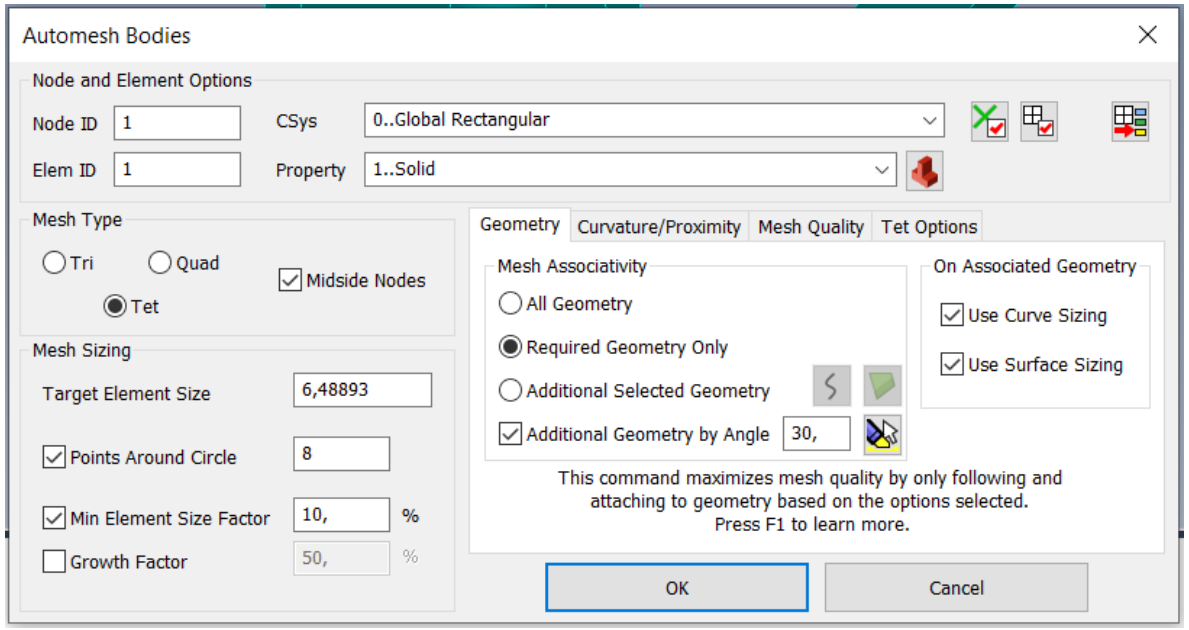
- **Target Element Size** will be computed to use for selected geometry based upon the **average** curve length of all edges in selected geometric entities divided by 6
- This parameter is used in **conjunction** with the other parameters in the **Mesh Sizing** section, along with the settings on the **Geometry**, **Curvature/Proximity**, and **Mesh Quality** to create either a triangular/quadrilateral surface or tetrahedral solid mesh
- When creating Tetrahedral Mesh on solids, Tetrahedral Mesher uses options specified on **Tet Options** tab

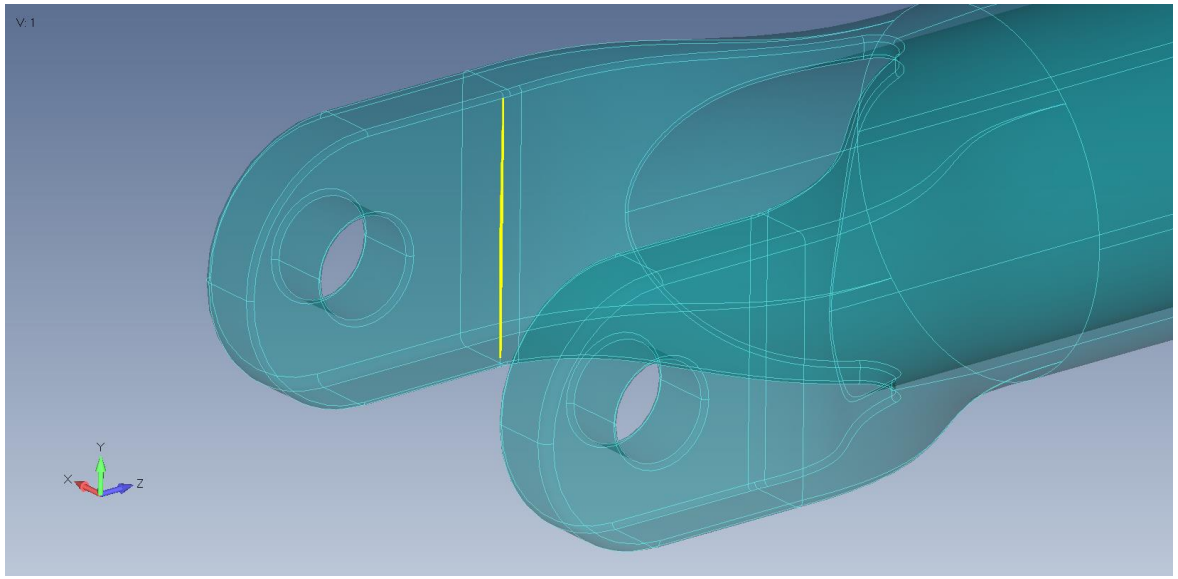


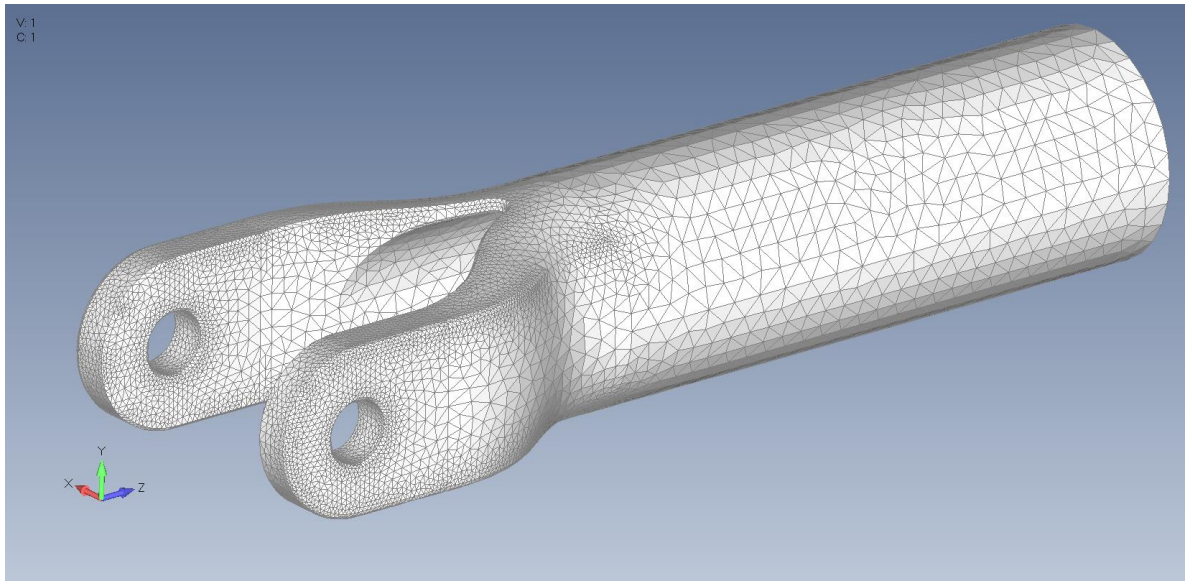
- **Mesh, Bodies** more details can be found in *Commands Manual, Chapter 5.1.4 Mesh, Bodies*

Body Mesher Technology
Suggested Workflow

- Using **Body Mesher** to automatically remove sliver surfaces and capture local curvature in this part



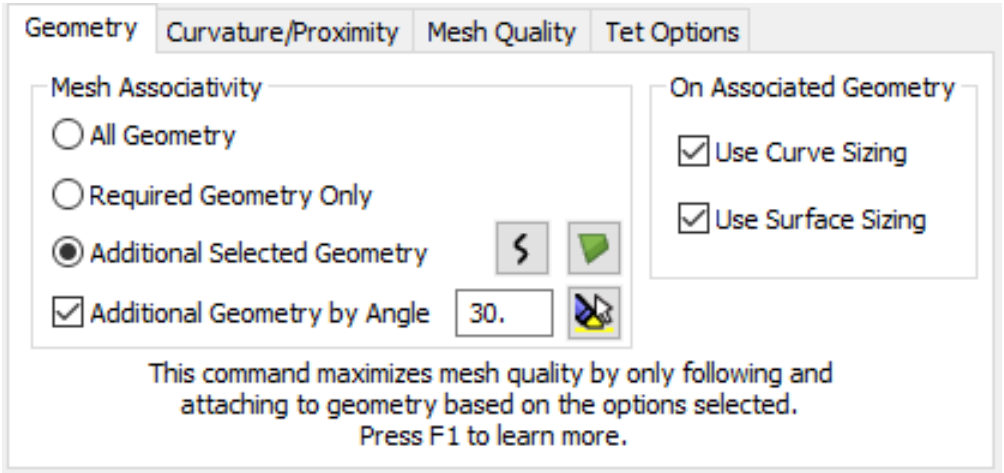


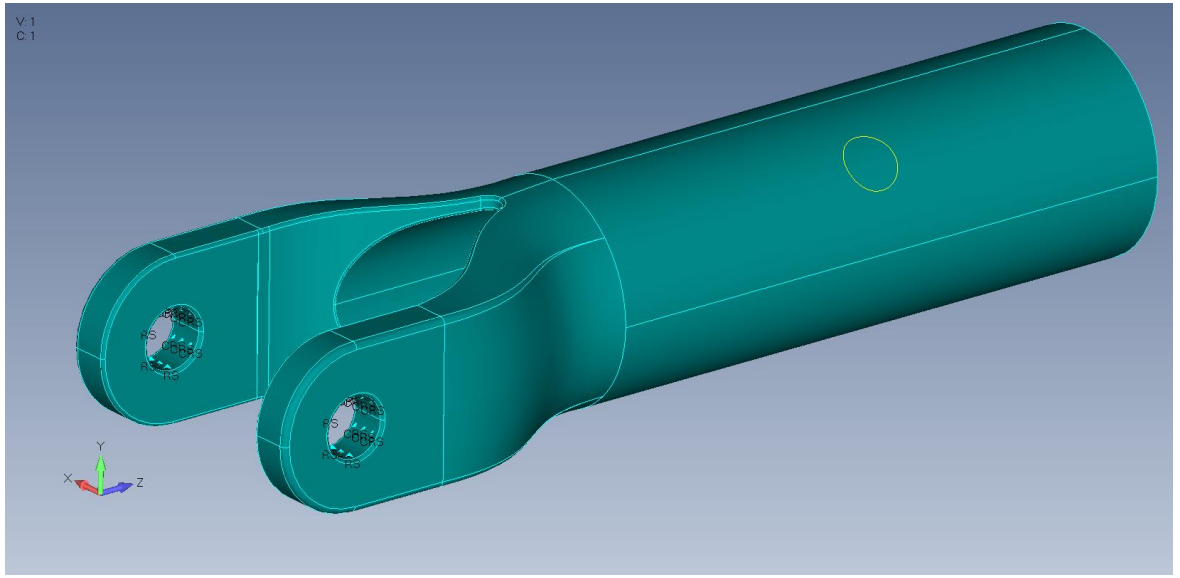


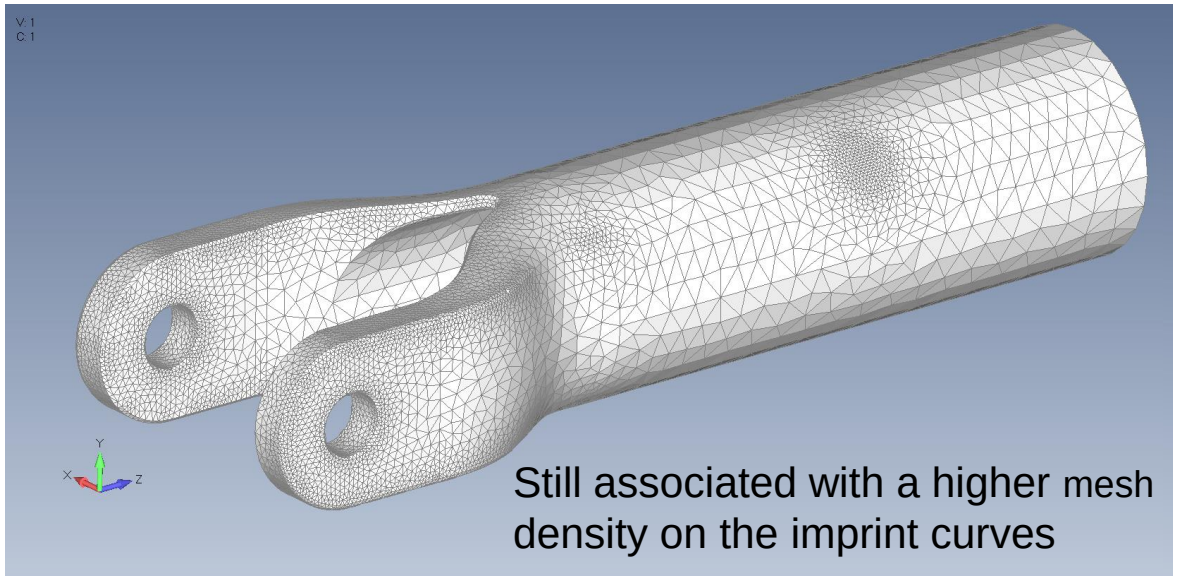
Body Mesher Technology
Suggested Workflow

FEMComp Engineering
Focusing On FEA

- We also need **associated geometry** for a **constraint** applied to the holes, and to the **circular imprint** for later connection to additional mesh



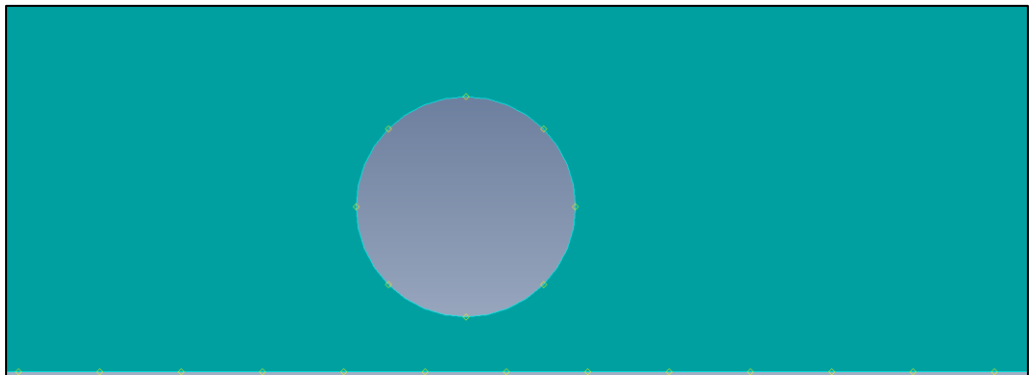




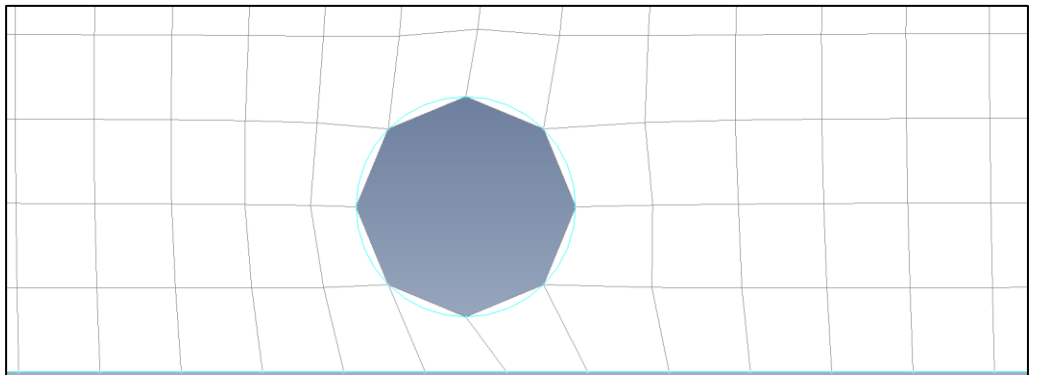
Workflow Change – specify Loads, Boundary Conditions, Contact Regions, etc., **BEFORE Body Meshing**!

SIEMENS

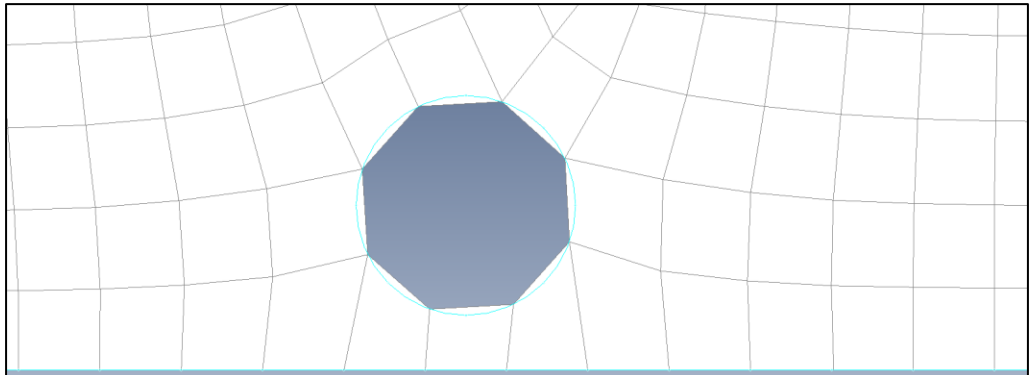
Body Mesher Technology
Mesh Sizing Example



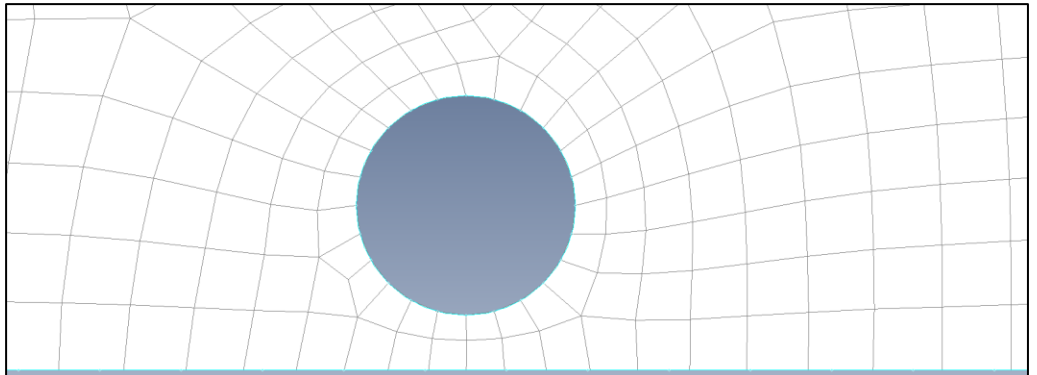
Traditional Enforced Mesh Locations



Forces **Poorly Shaped** Quads



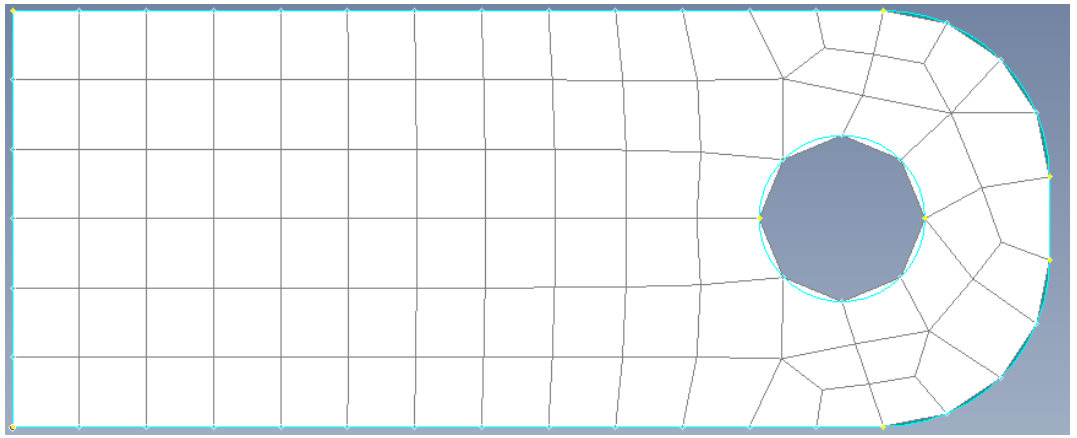
Body Meshing Target **Size Adjusts**
To Maximize Element Quality



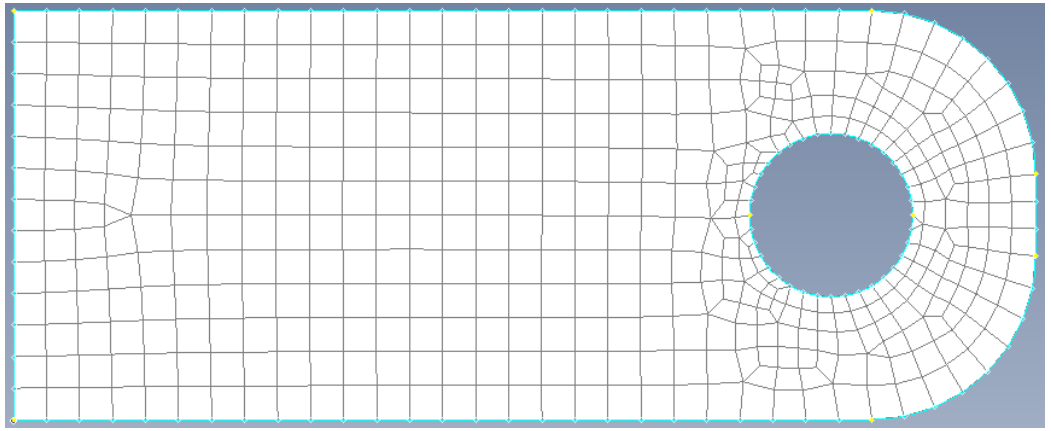
Smaller Target Size on interior curves
adjusts even more

Body Mesher Technology
Mesh Sizing Example

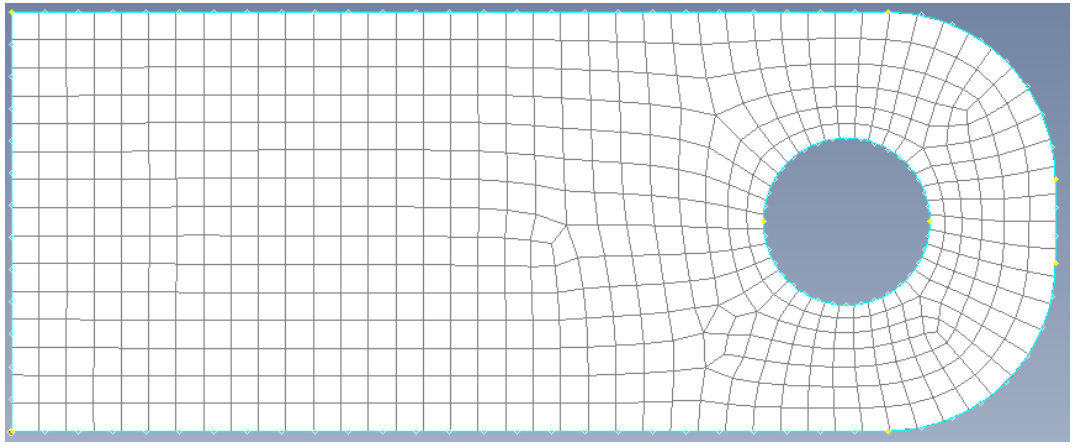
Traditional Enforced Mesh Locations



Default **Poorly Shaped** Quads



Traditional Mesh with **Mesh Size refinement**

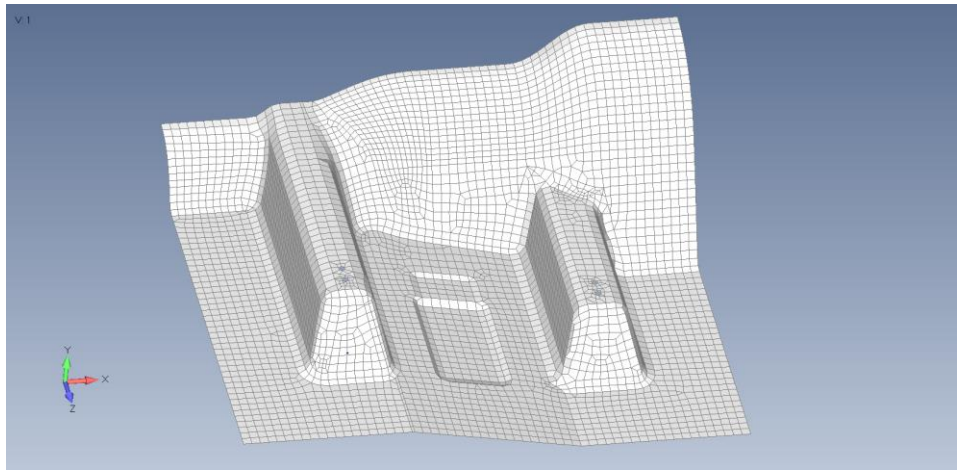


Body Meshing with **Target Size** equal to Mesh Sizes from Traditional Mesh

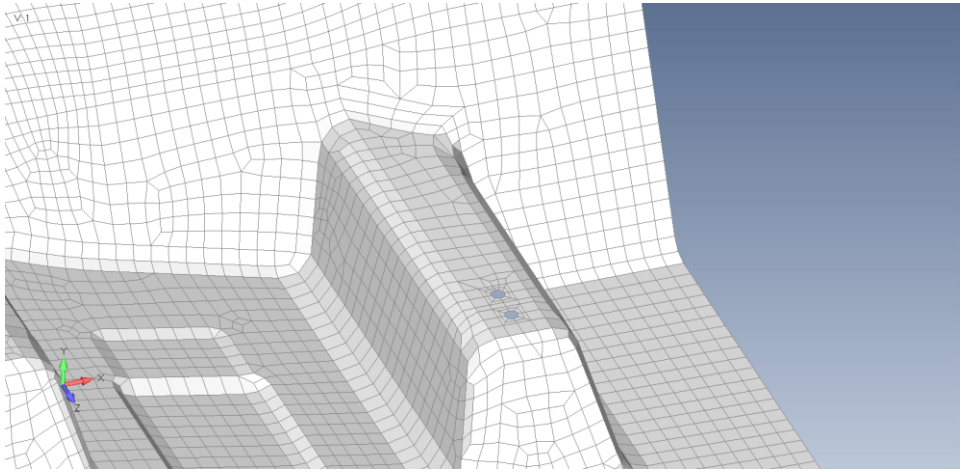
Body Mesher Technology
Body Meshing Cleanup Example

FEMComp Engineering
Focusing On FEA

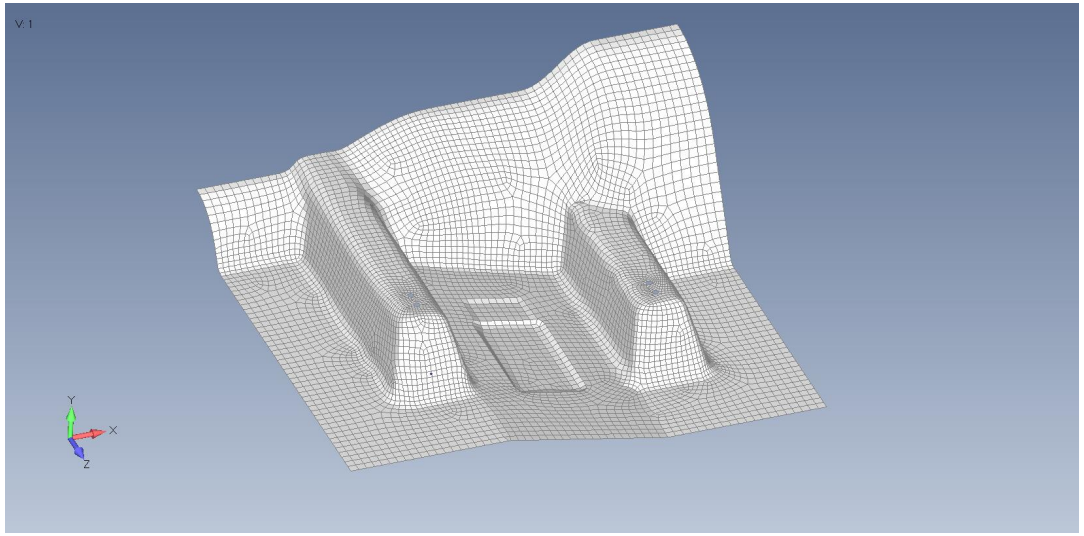
Automatic Geometry Preparation and Mesh



Manual Geometry Cleanup and Mesh



With **Body Meshing -** set **Target Size** and let the Mesher take care of everything else



SIEMENS

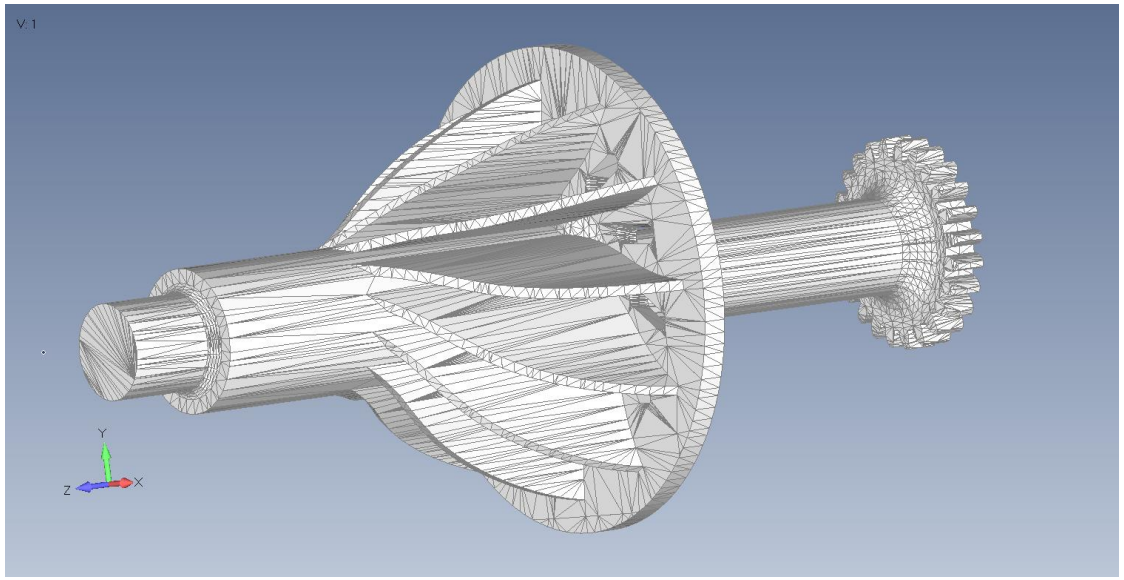
Body Mesher Technology
Body Meshing Benefits

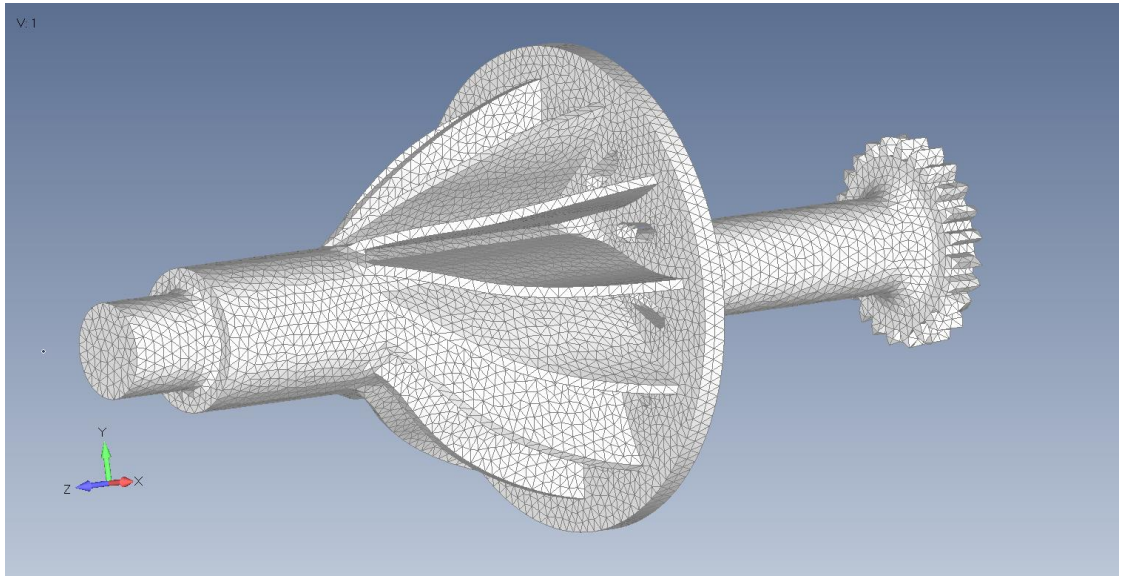
Since triangular facets are all that are required for meshing, **new possibilities** are now available in **Femap**

“Clean” STL files can be used as the basis to create a high-quality surface mesh

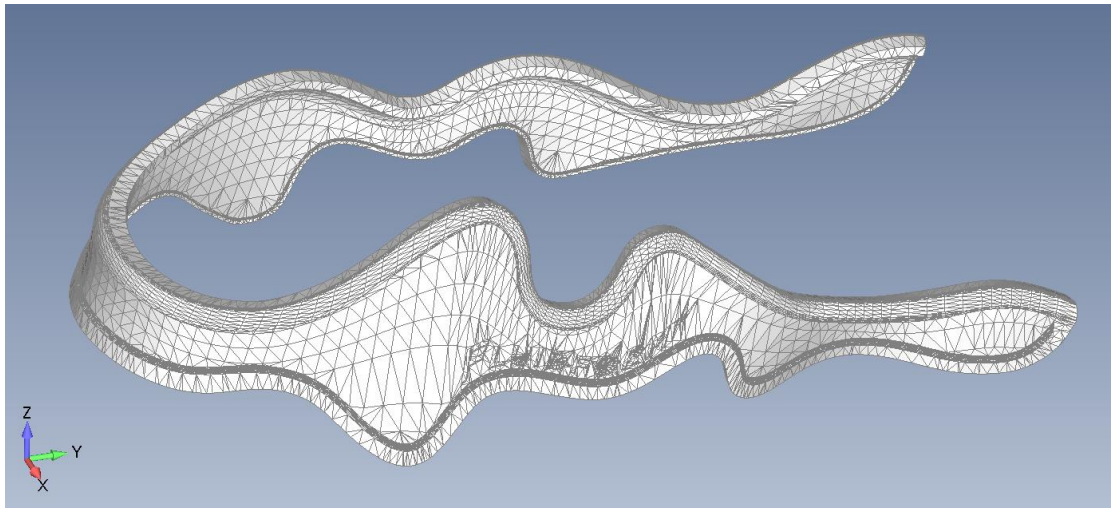
Note: “Clean” STL files have:

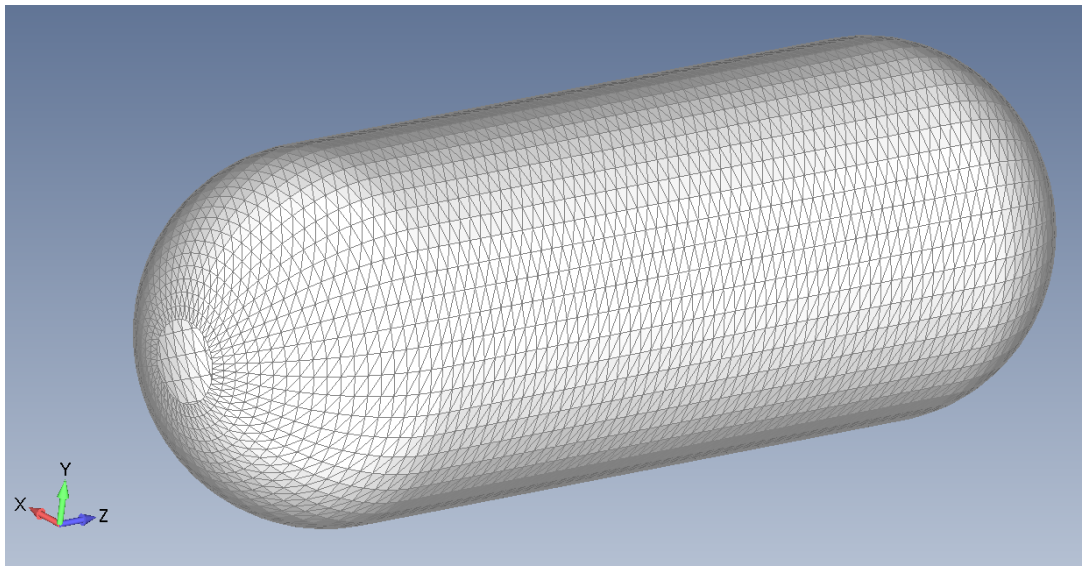
- All facets fully connected
- No facets which overlap

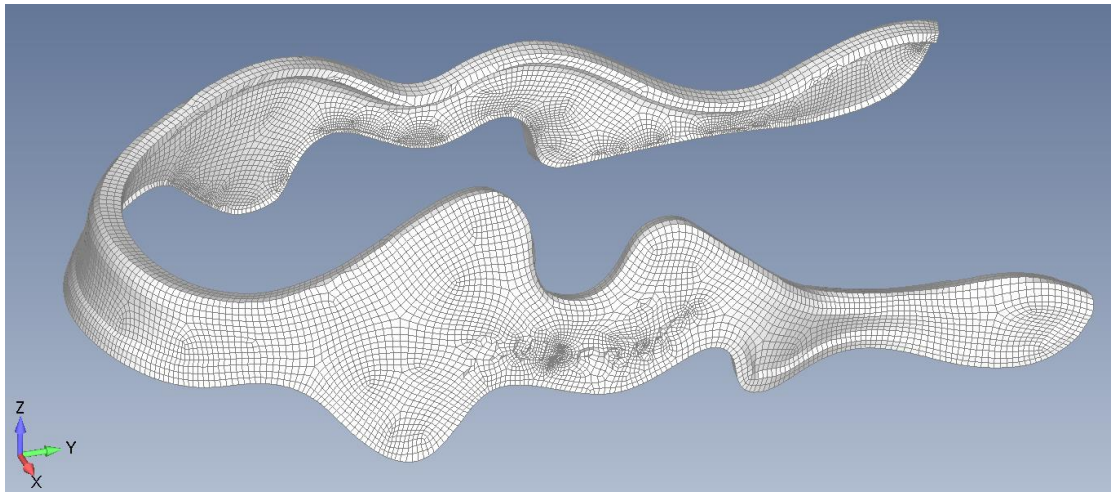


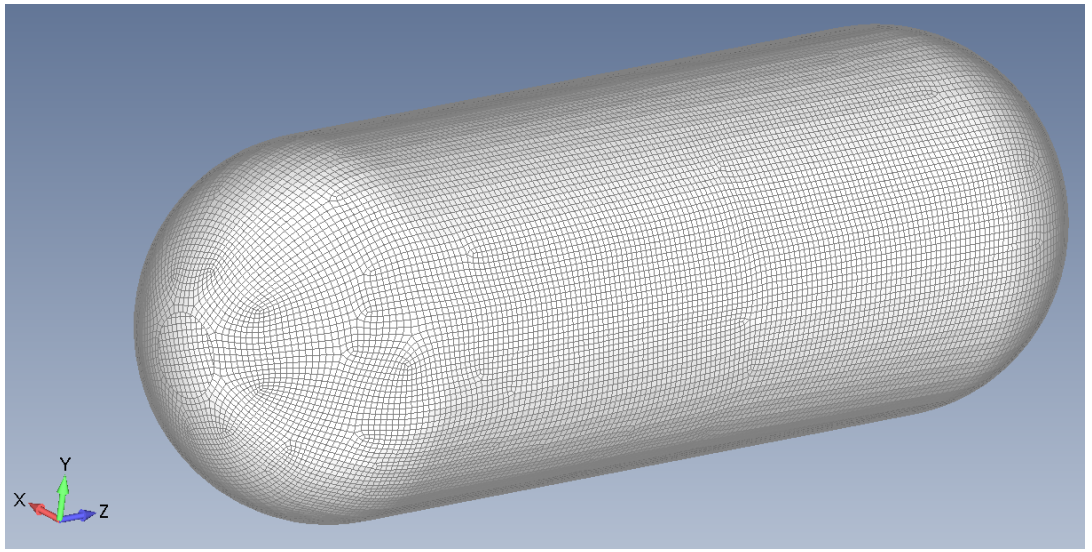


Body Mesher Technology
STL Surface Meshing Examples







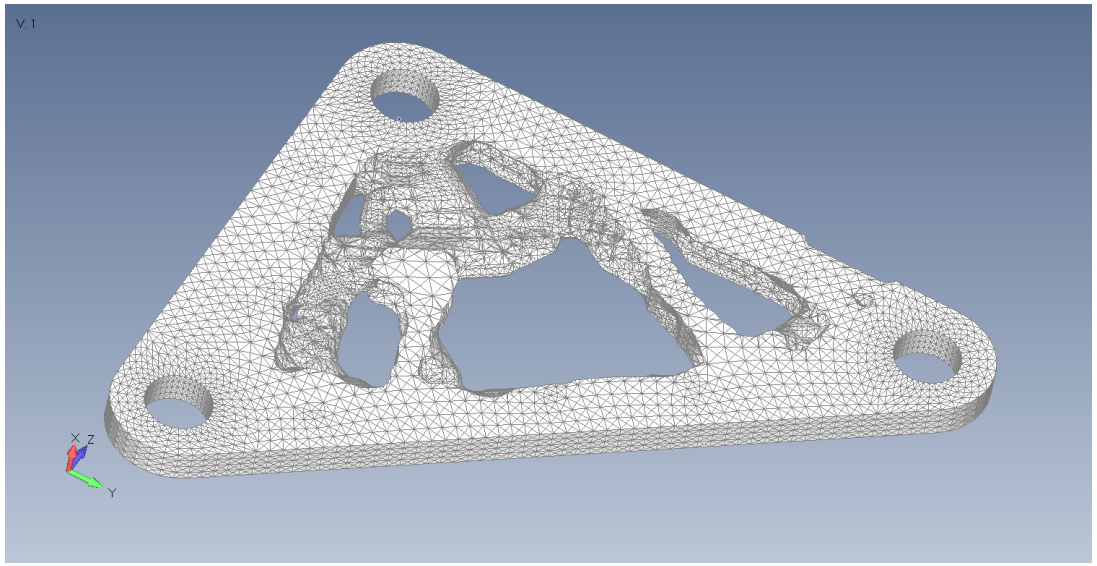


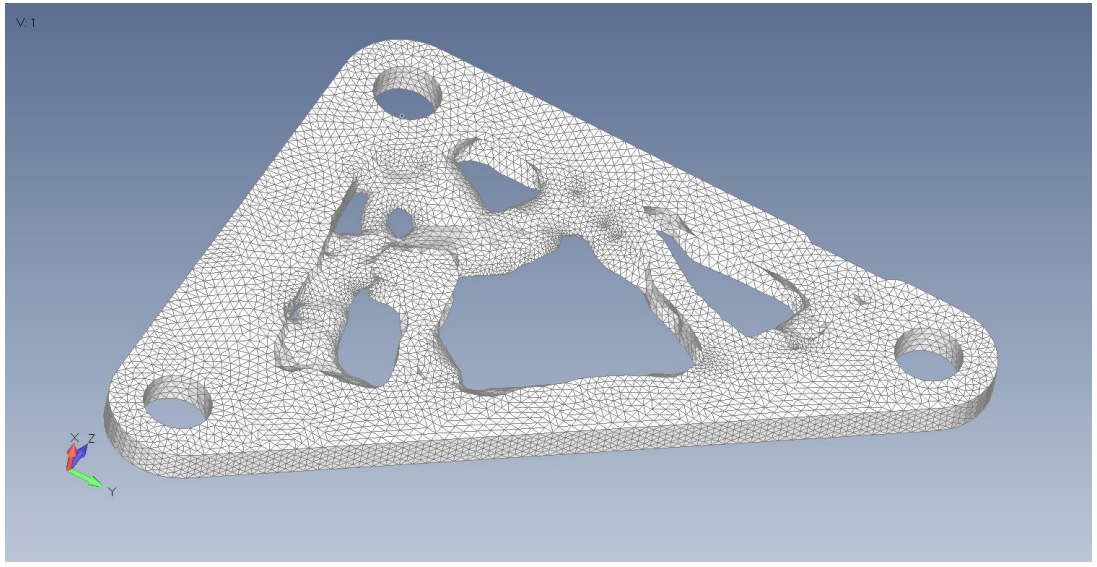
Body Mesher Technology
Body Meshing Benefits

FEMComp Engineering
Focusing On FEA

Since triangular facets are all that are required for meshing, **new possibilities** are now available in **Femap**

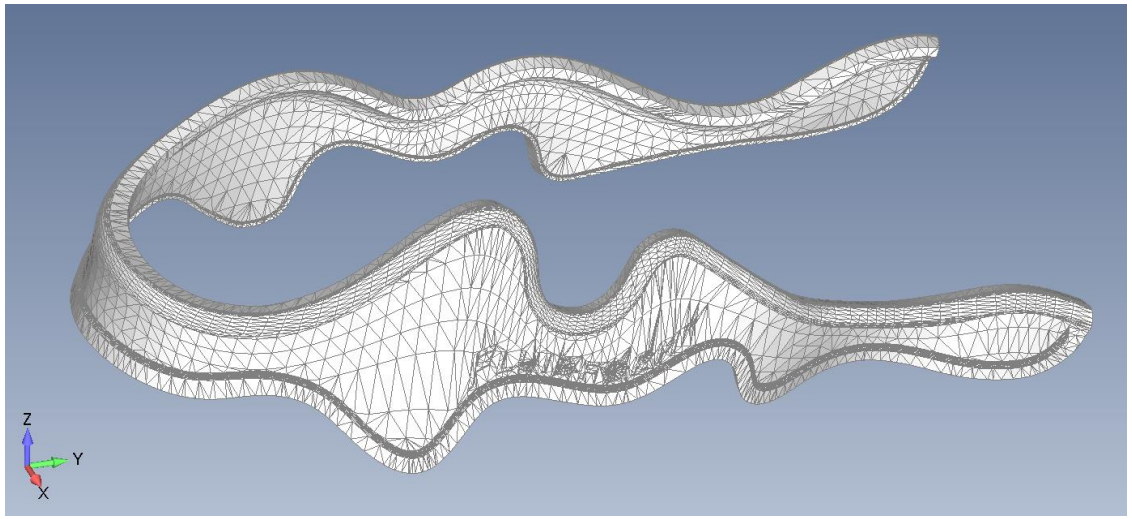
STL facets of topology optimized shapes can be turned into **useful tetrahedral meshes**

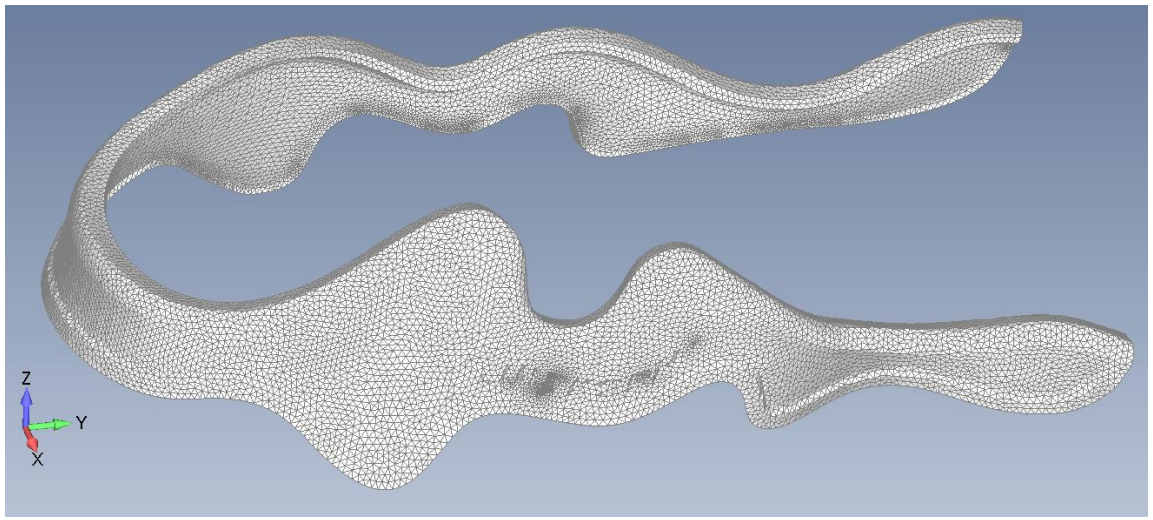


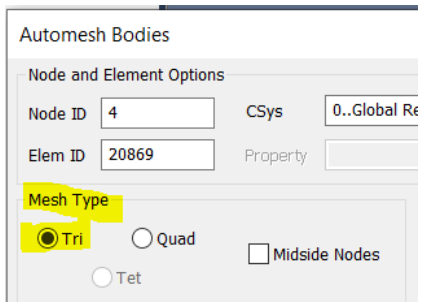


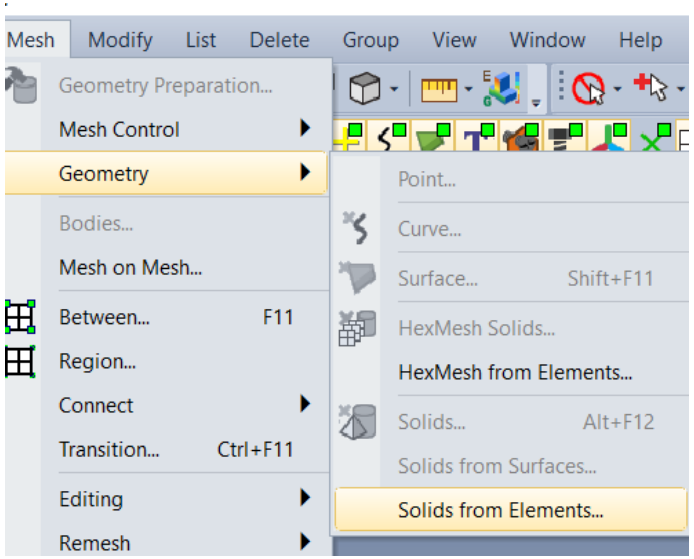
SIEMENS

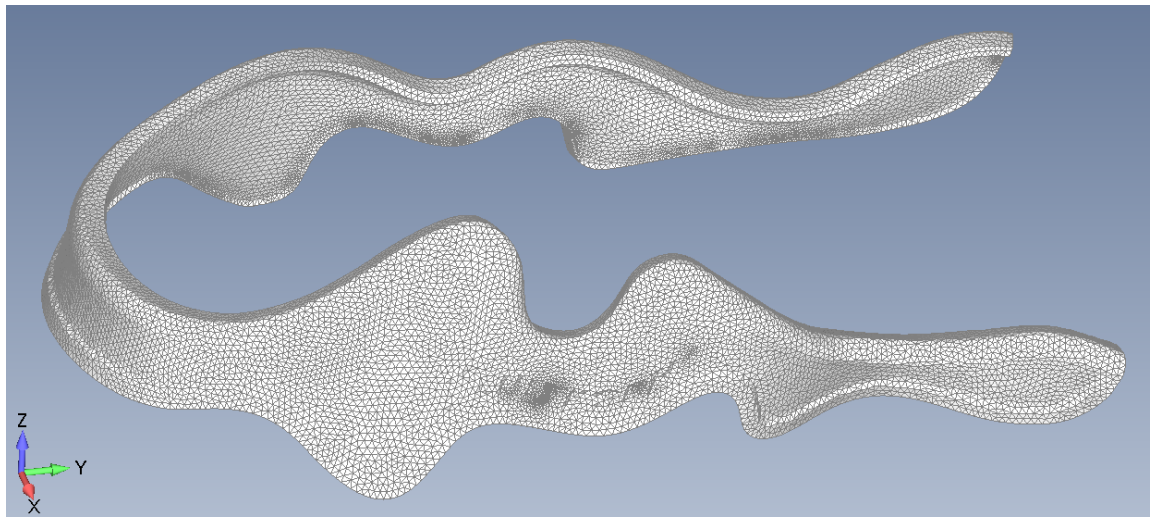
Body Mesher Technology
STL Solid Meshing Example











Body Mesher Technology
Body Meshing Benefits

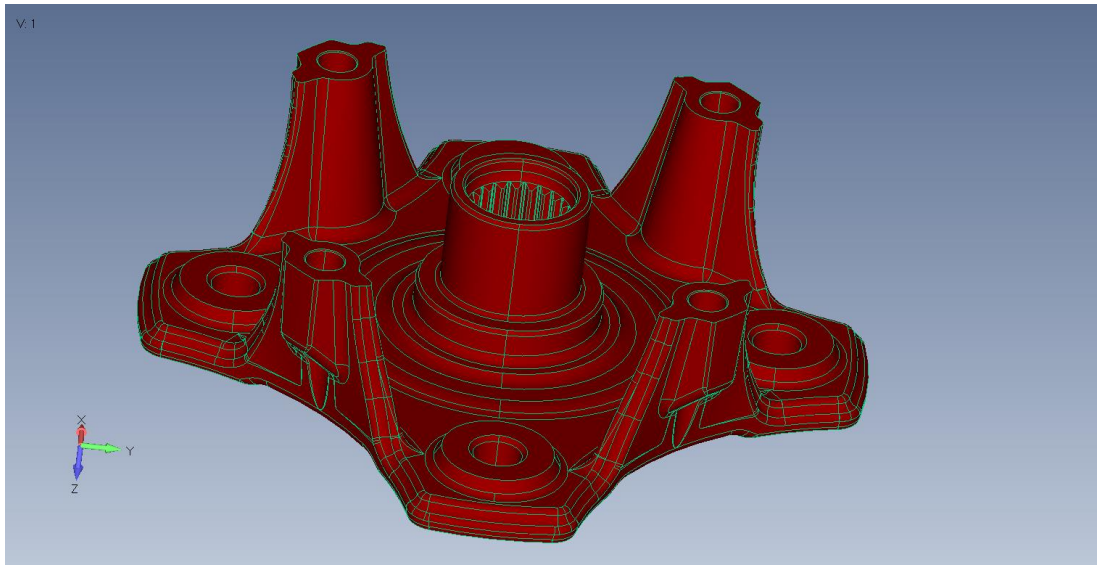
Automatic Cleanup and **Defeaturing** for **Tetrahedral** Meshing

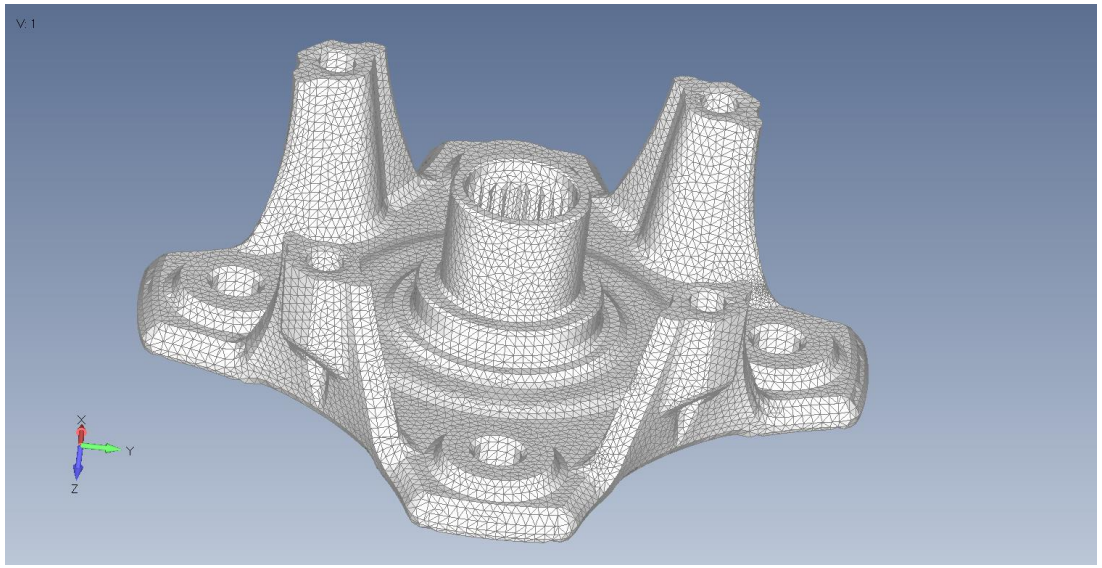
To mesh this part in previous versions:

- First, use the “Mesh, **Geometry Preparation**” command
- Second, use various tools in the Meshing Toolbox to create the desired triangular **surface mesh**
- Finally, use “Mesh, Geometry, **Solids from Elements**” command to create Tet Mesh

To mesh this part **in 2021.2** and above

- Simply use “**Mesh, Bodies**” command





Body Mesher Technology
Body Meshing Example

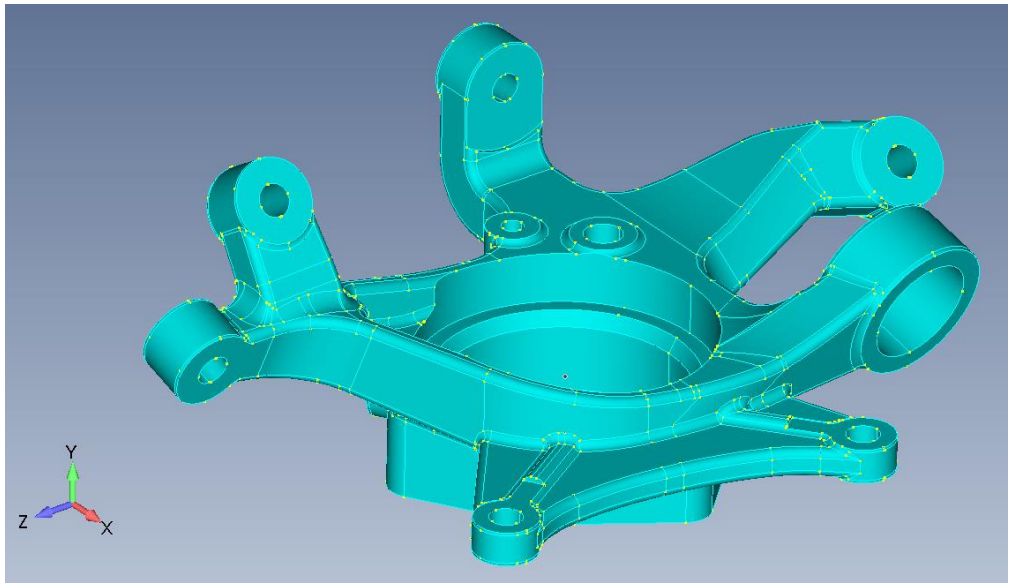
Automatic Cleanup and **Defeaturing** for **Tetrahedral** Meshing

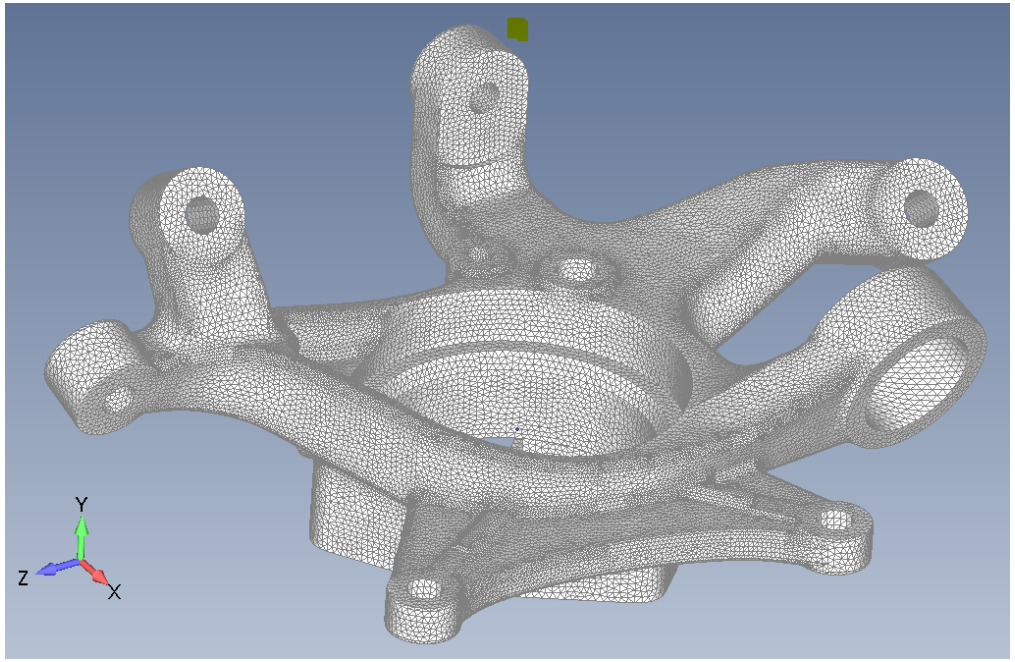
Traditional mesh of this part

- Mesh, **Geometry Preparation** command
- Triangular **surface mesh**
- Mesh, Geometry, **Solids from Elements**" command

Body Meshing **in 2021.2**

- Use **Mesh, Bodies** command

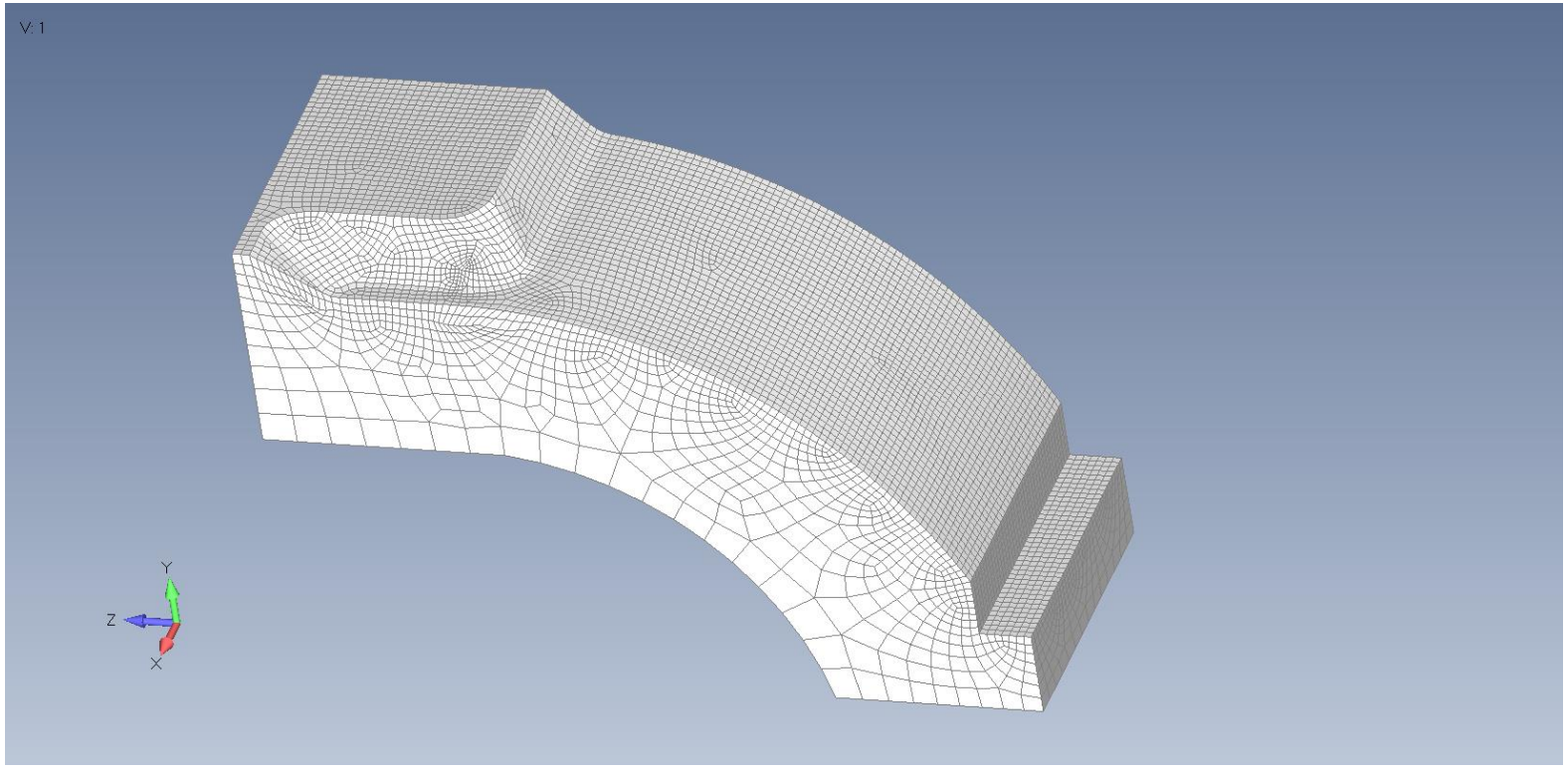




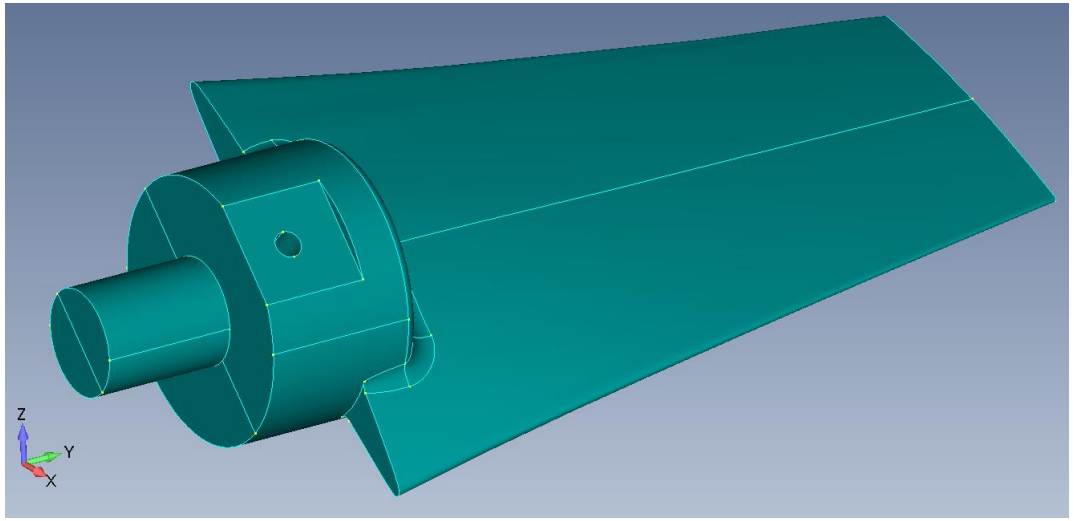
Body Mesher Technology
Powerful Transition Meshing

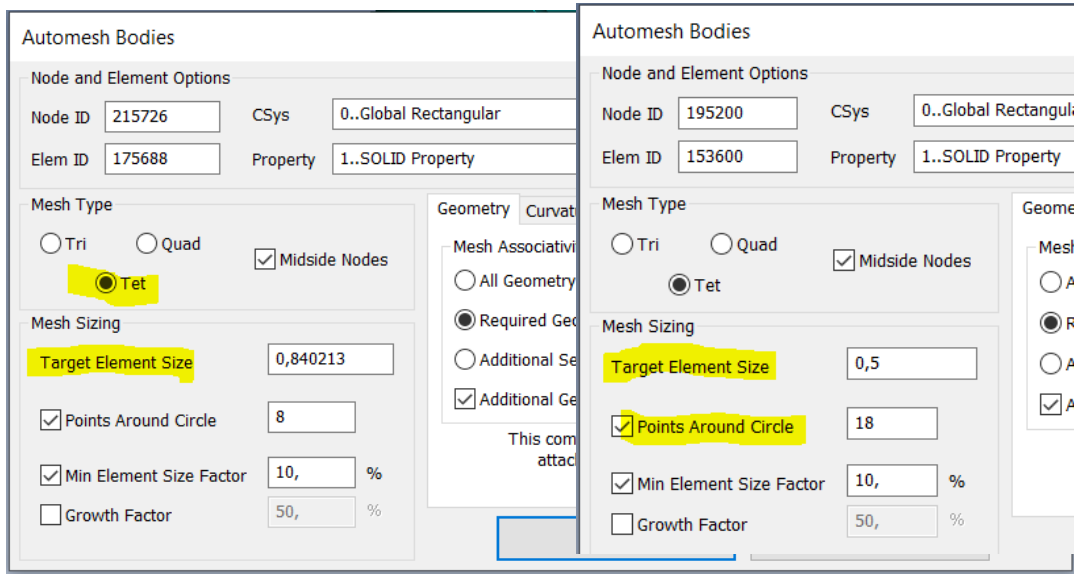
Triangle and **Quadrilateral-dominant** meshing via the **Body Mesher** also produces high-quality elements in **transition areas** where the element size changes to a large degree

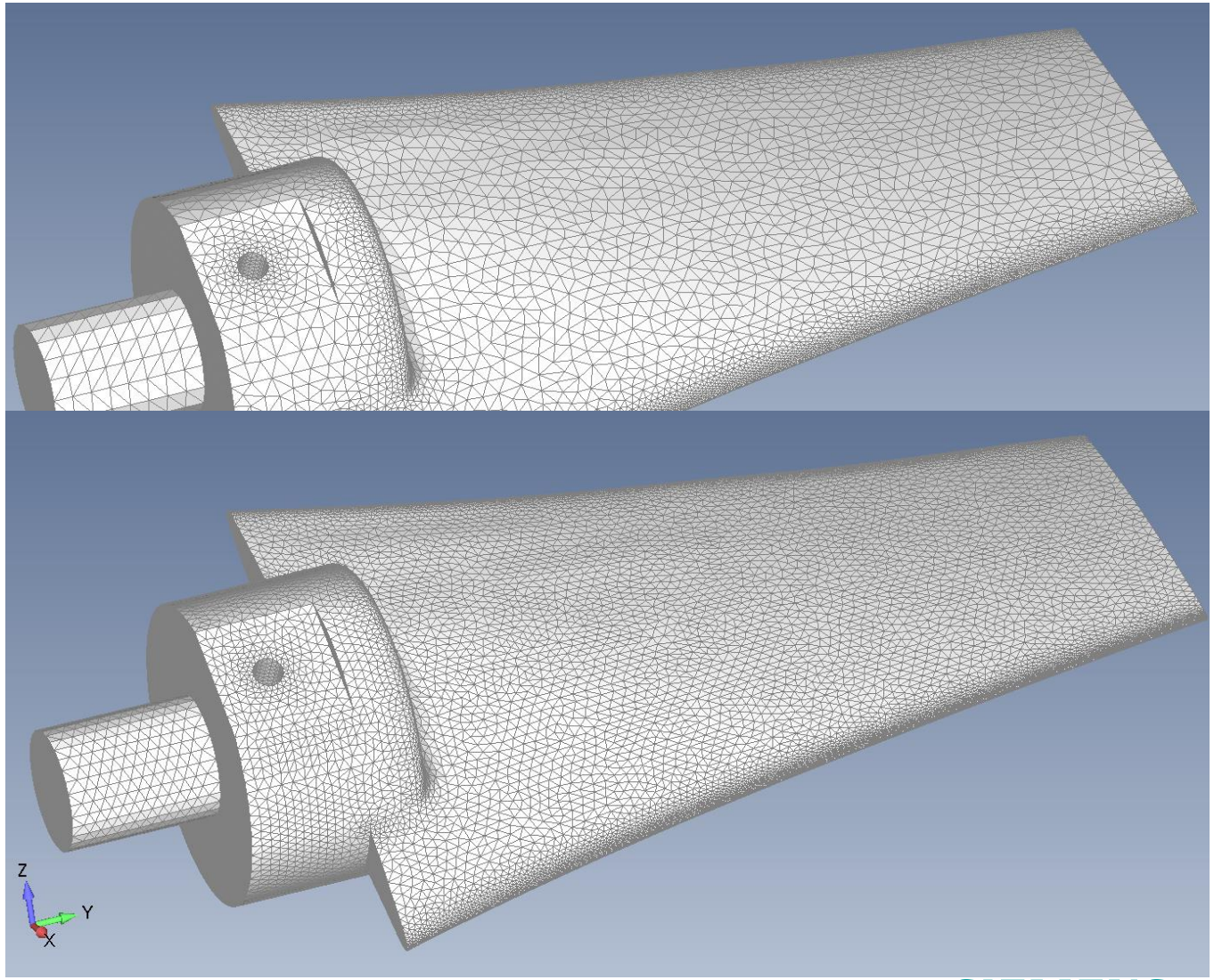
- Example shows **high density mesh enforced** on the surfaces shown which **transition** to larger elements **based** on the overall target **mesh size** set for the rest of the part



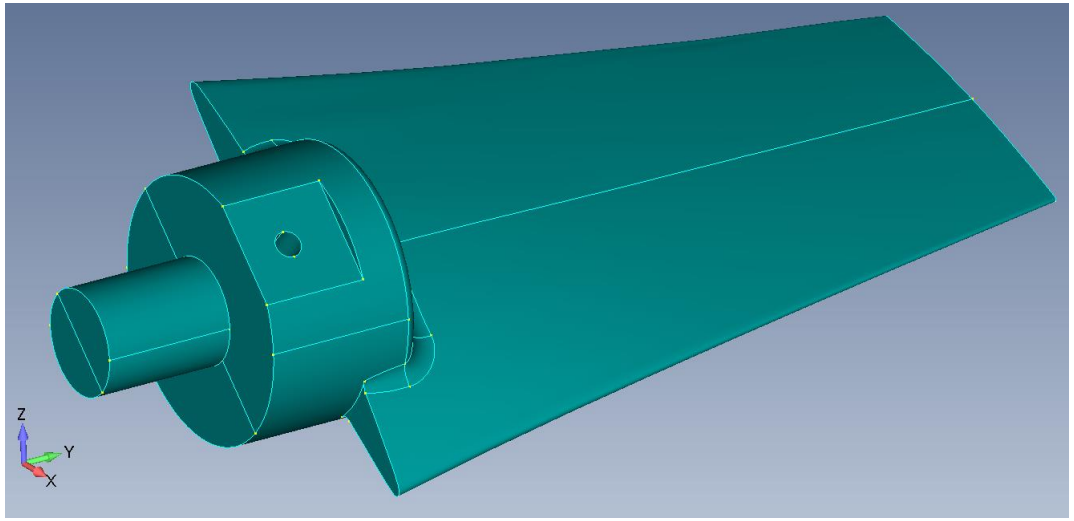
Body Mesher Technology
Mesh Bodies, Tet Solid Meshing Example

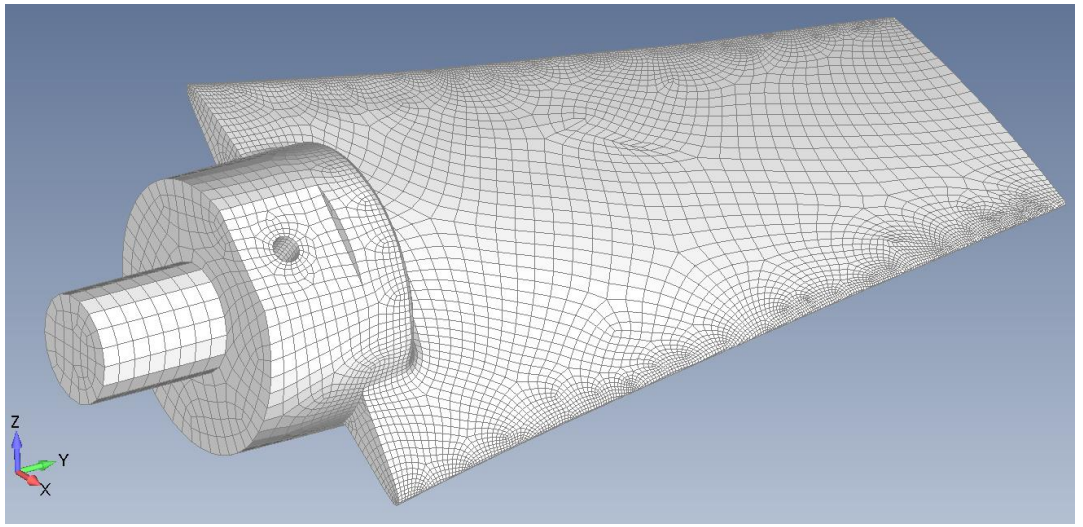


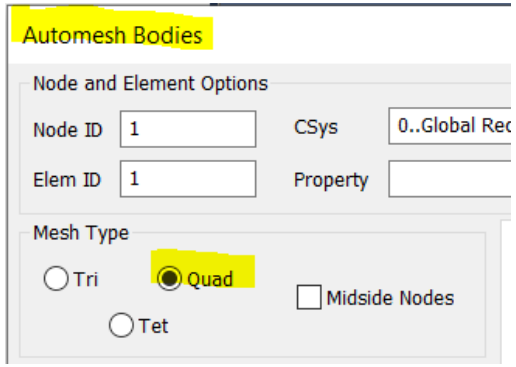


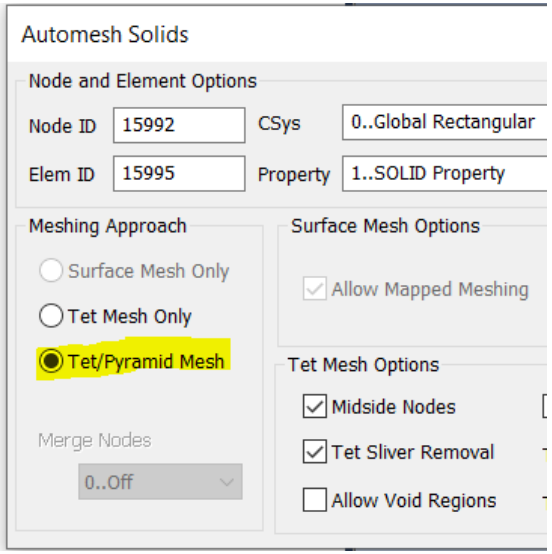


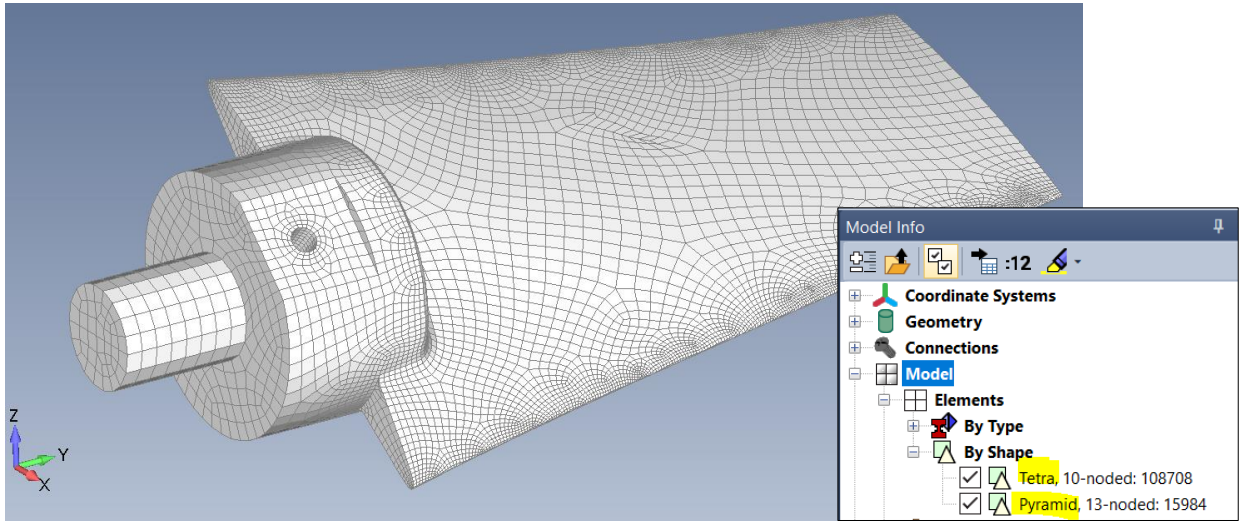
Body Mesher Technology
Quad Surface and Tet/Pyramid Solid Meshing Example







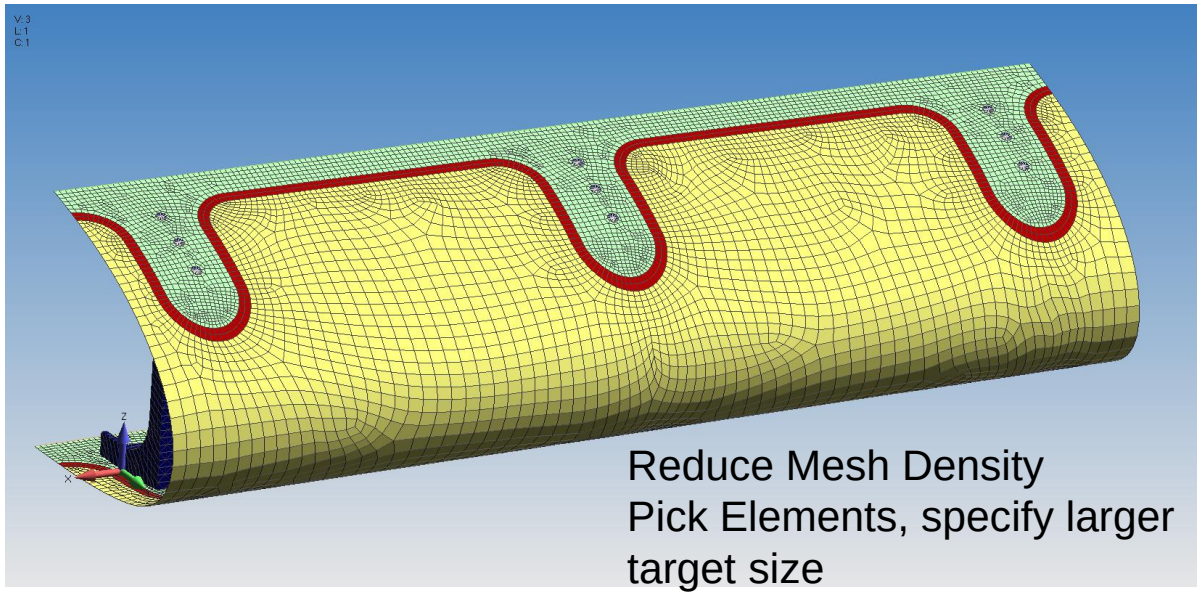


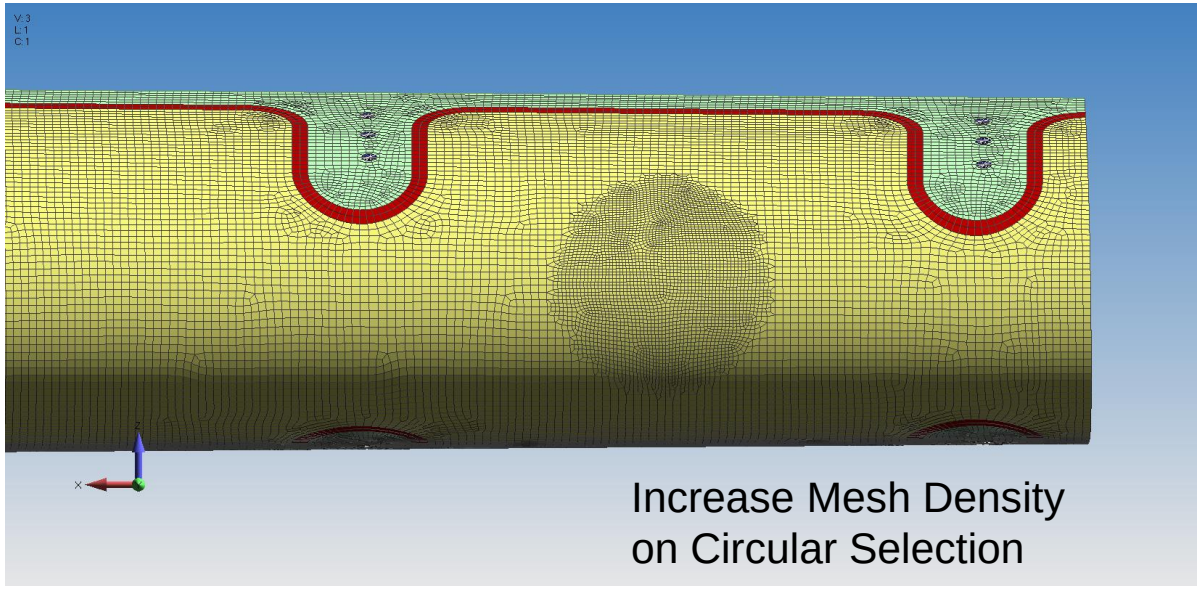


Body Mesher Technology
Mesh on Mesh

New **Mesh, Mesh on Mesh** command offers the ability to:

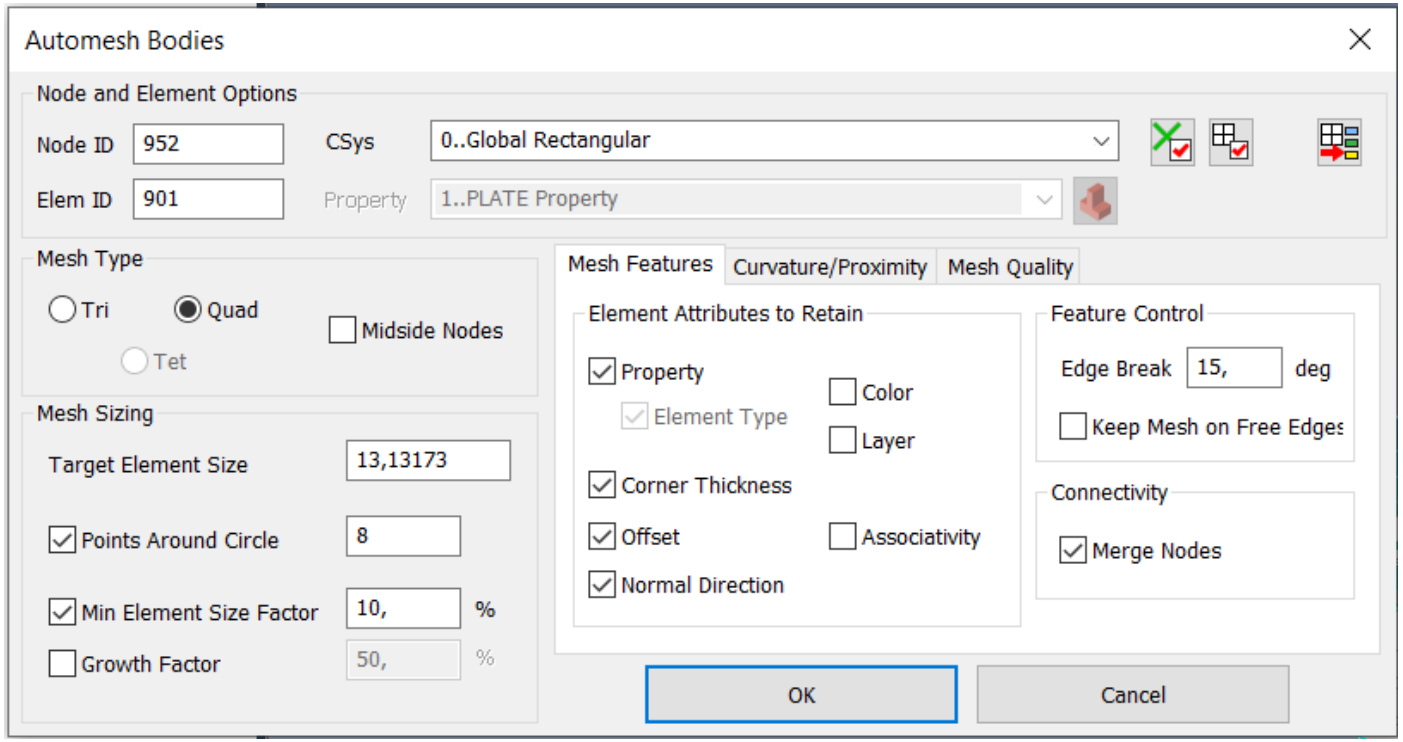
- **Convert** any triangulation generated from an existing mesh into a **high-quality triangle** or **quadrilateral-dominant** mesh
- **Specify** mesh sizing and surface curvature/proximity options
- **Retain** selected features of original mesh
- **Create elements** which attempt to satisfy user-defined element quality criteria





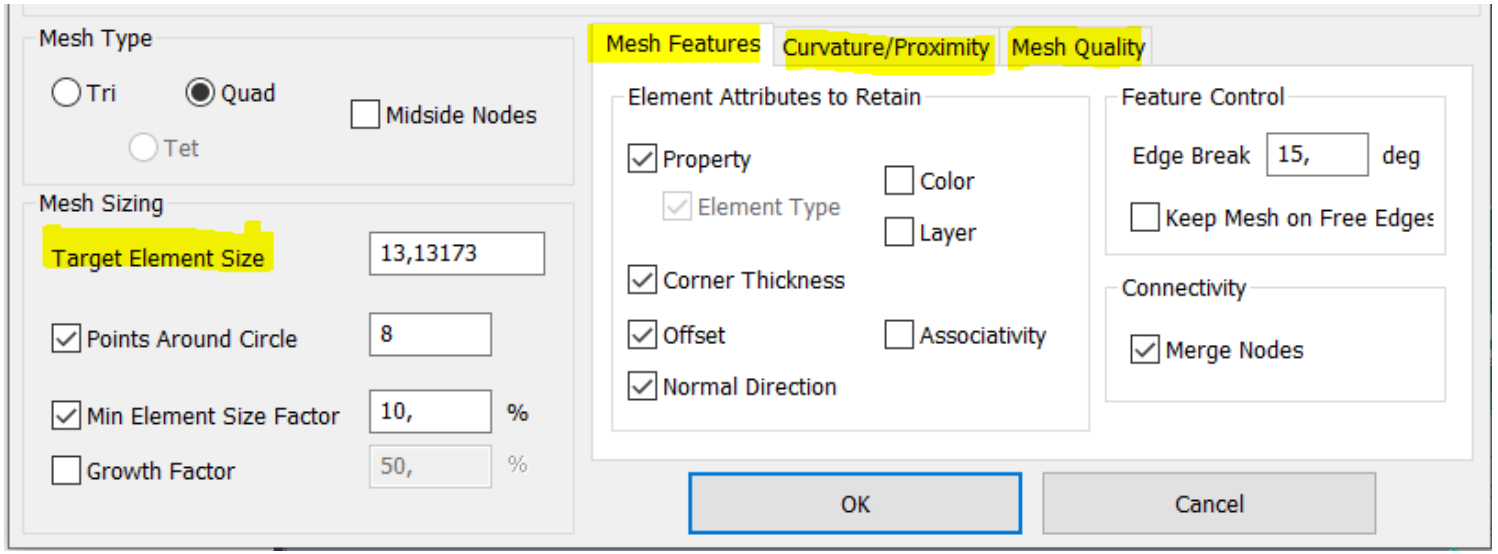
Body Mesher Technology
Mesh, Mesh on Mesh…

- Uses meshing methodologies as **Mesh, Bodies** command, only it uses **existing** triangular or quadrilateral elements as the starting point
- One other difference is this command **cannot** create **Tetrahedral** elements
- The elements can be **imported** or **built** directly inside of FEMAP
- Once the elements exist in FEMAP, they are selected using the standard entity selection dialog box



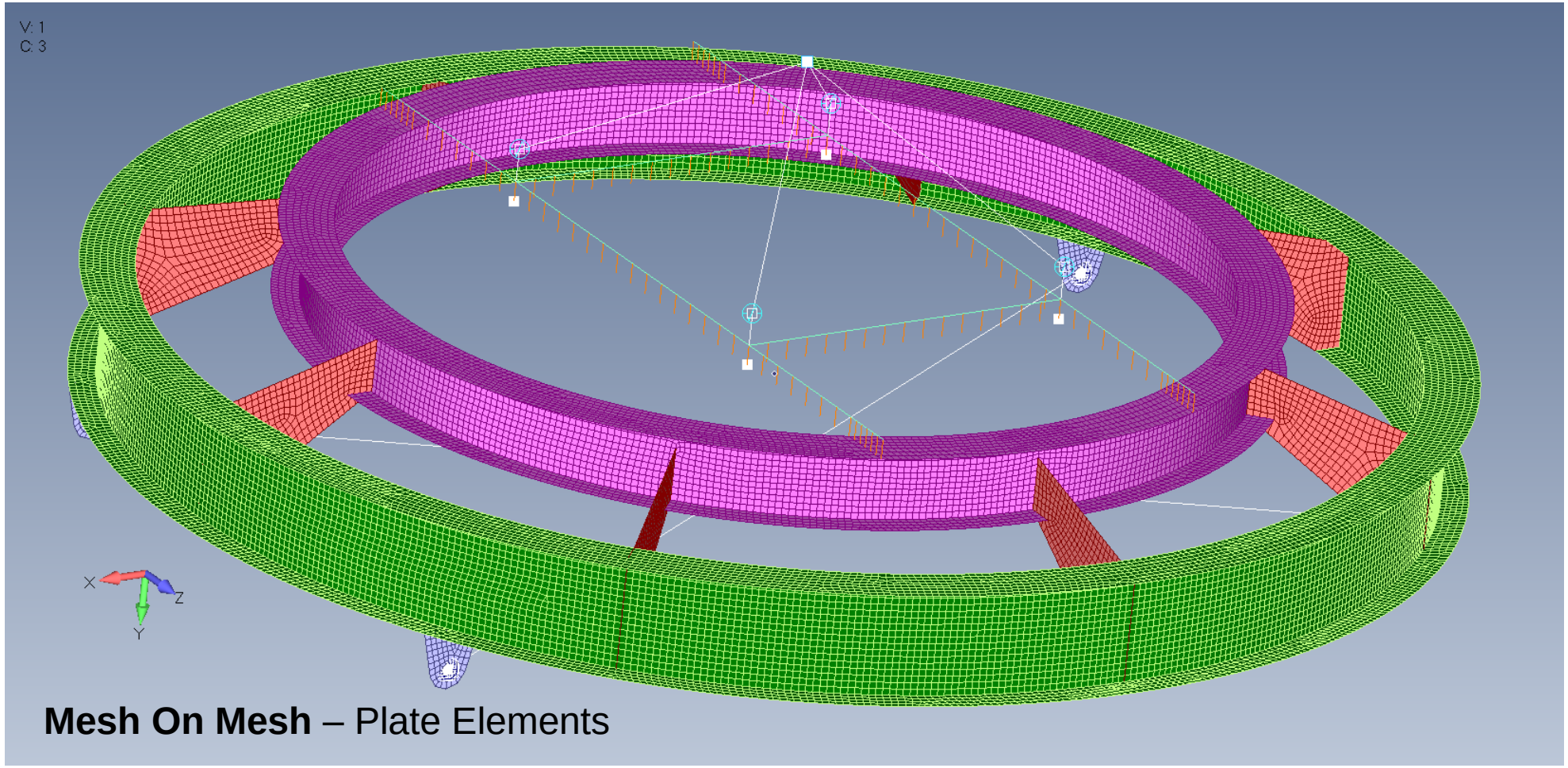
Body Mesher Technology
Mesh, Mesh on Mesh…

- Once sections of elements have been selected, **Automesh Bodies** dialog box will be displayed
- **Target Element Size** will be computed to use for the selected elements based upon the average **edge length** of the selected **elements**
- This parameter is used in **conjunction** with the other parameters in the **Mesh Sizing** section, along with the settings on the **Mesh Features**, **Curvature/Proximity**, and **Mesh Quality** to create either a triangular/quadrilateral mesh which will replace an existing mesh

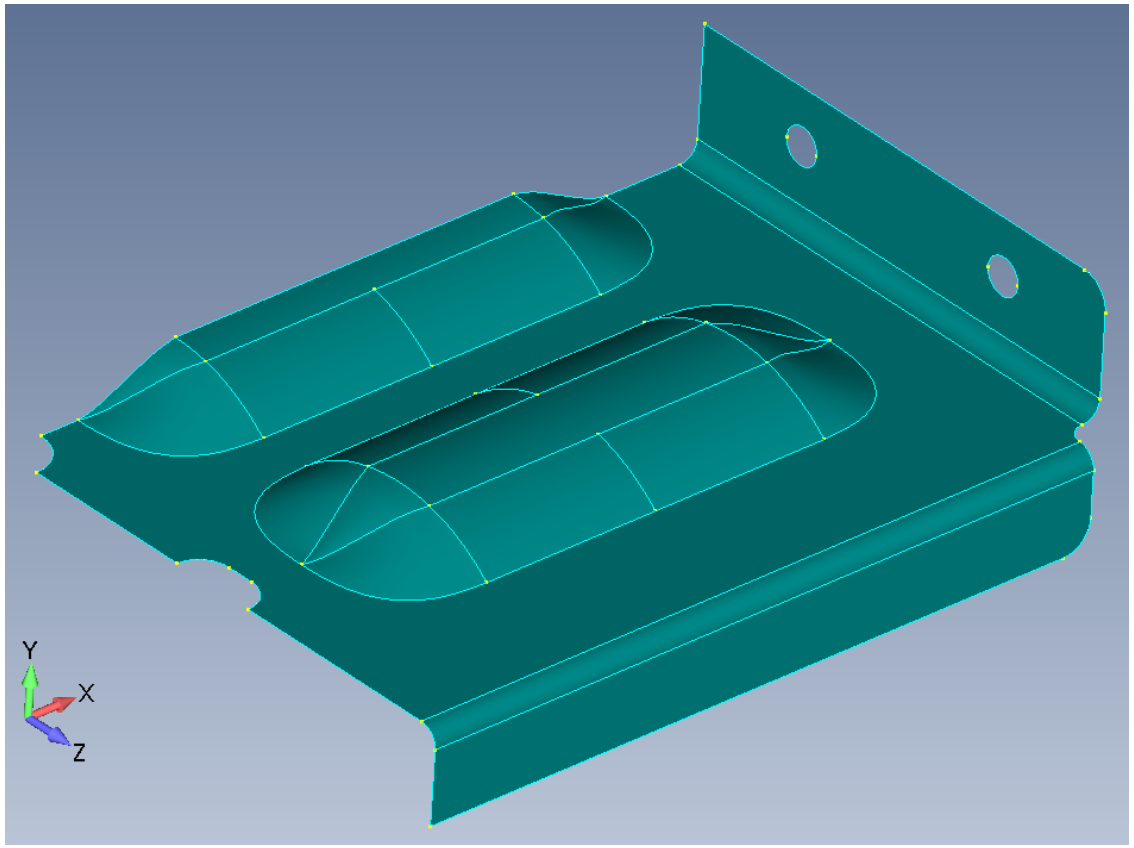


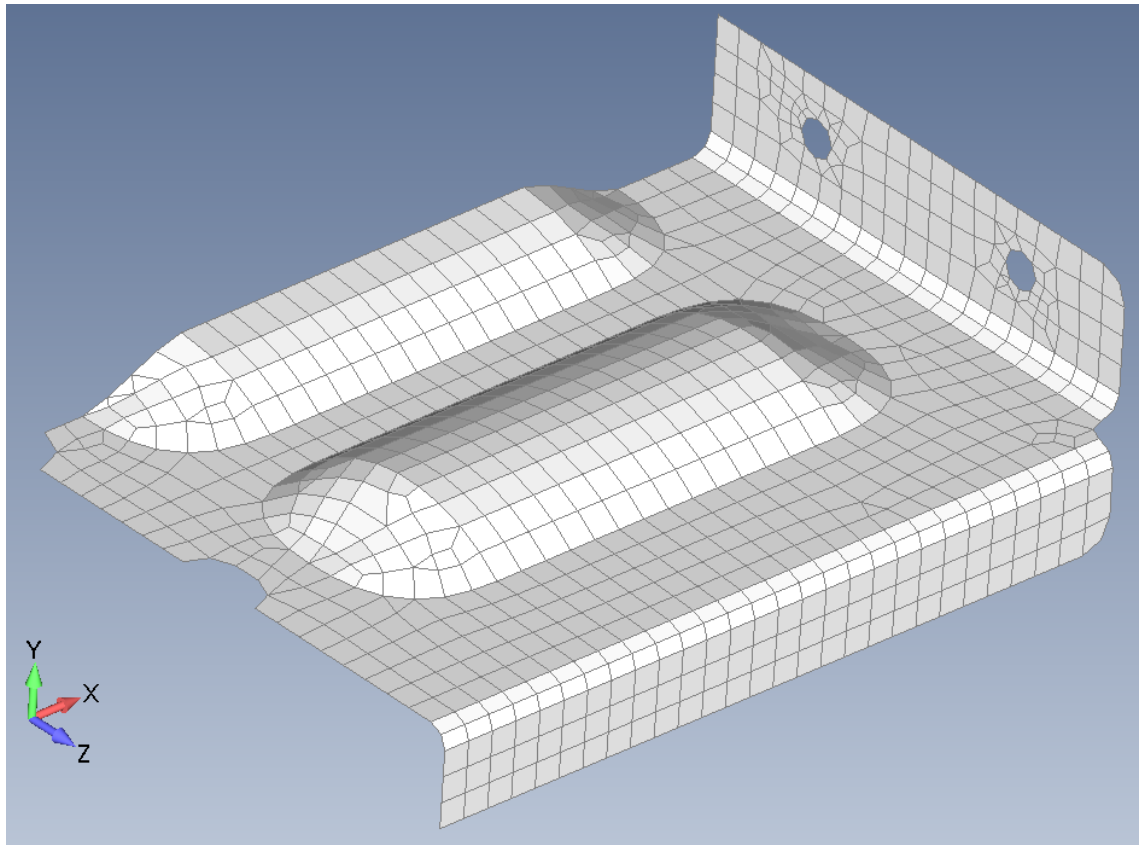
- **Mesh, Mesh on Mesh…** more details can be found in *Commands Manual, Chapter 5.1.5 Mesh, Mesh on Mesh…*

Body Mesher Technology
Mesh on Mesh – Mesh Refinement



Body Mesher Technology
Mesh on Mesh Example



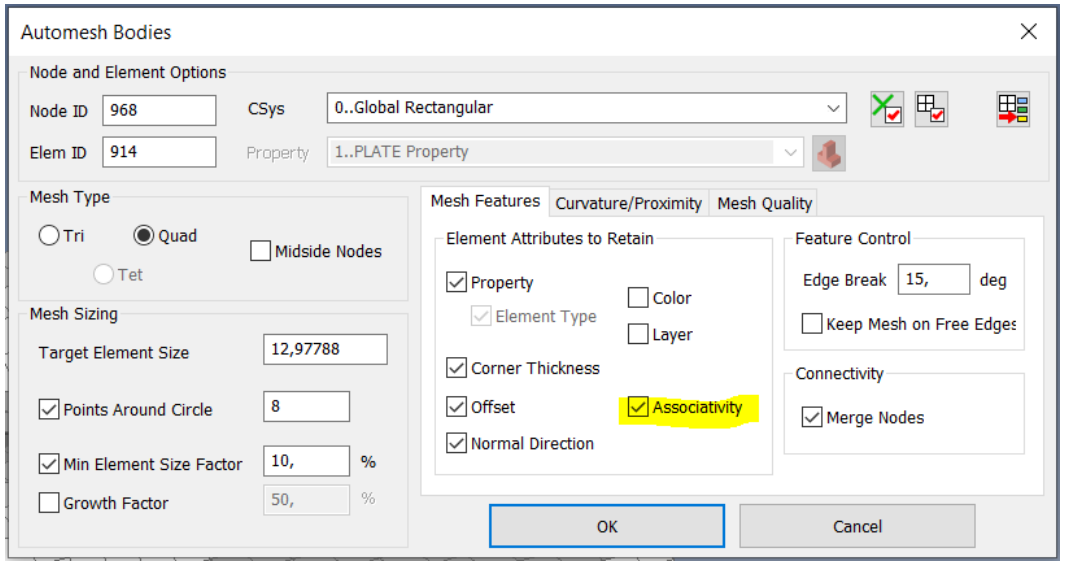


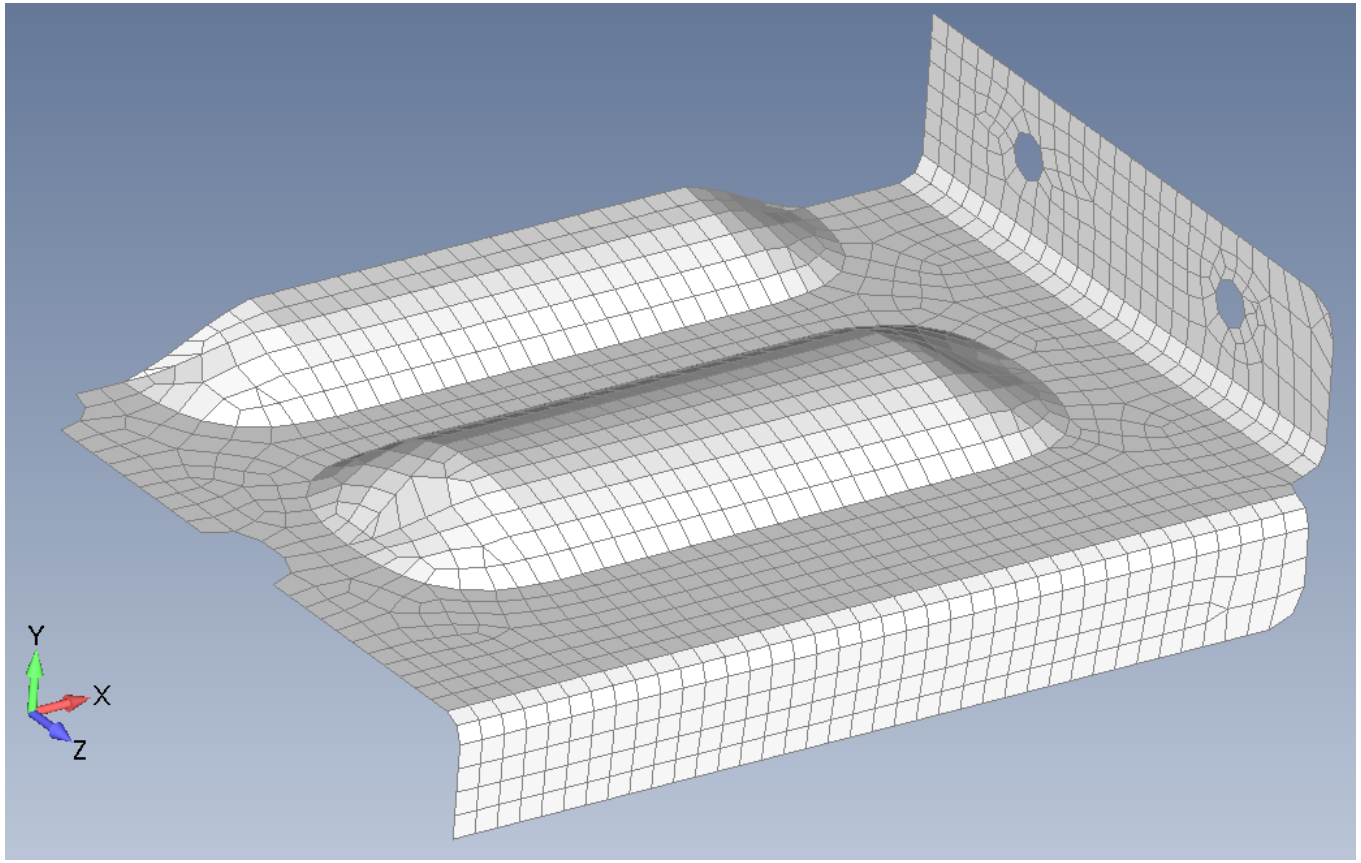
Initial Mesh using **Mesh, Geometry, Surface…**

Body Mesher Technology
Mesh on Mesh Example

FEMComp Engineering
Focusing On FEA

Mesh On Mesh – initial Target Element Size for whole model with **Associativity** to geometry



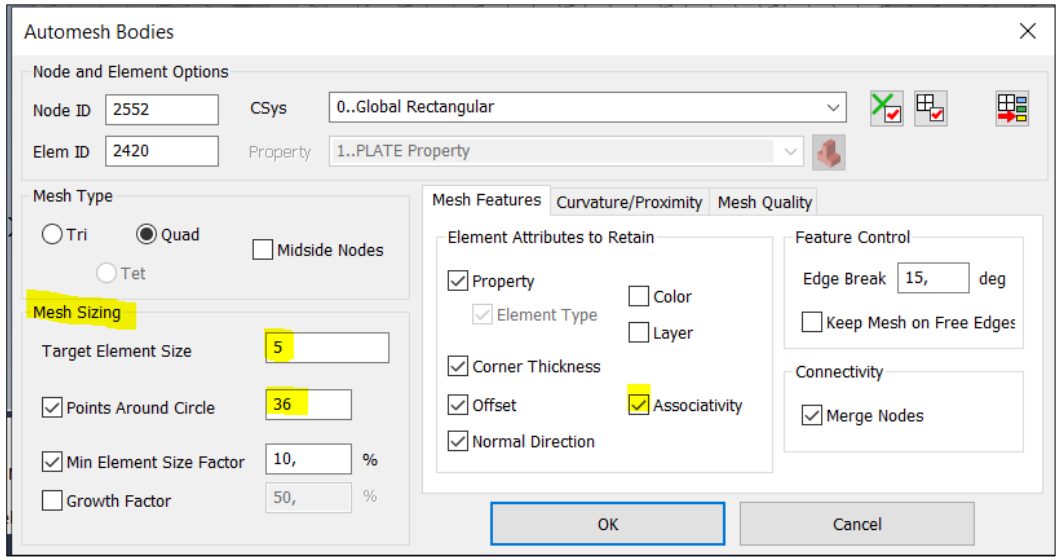


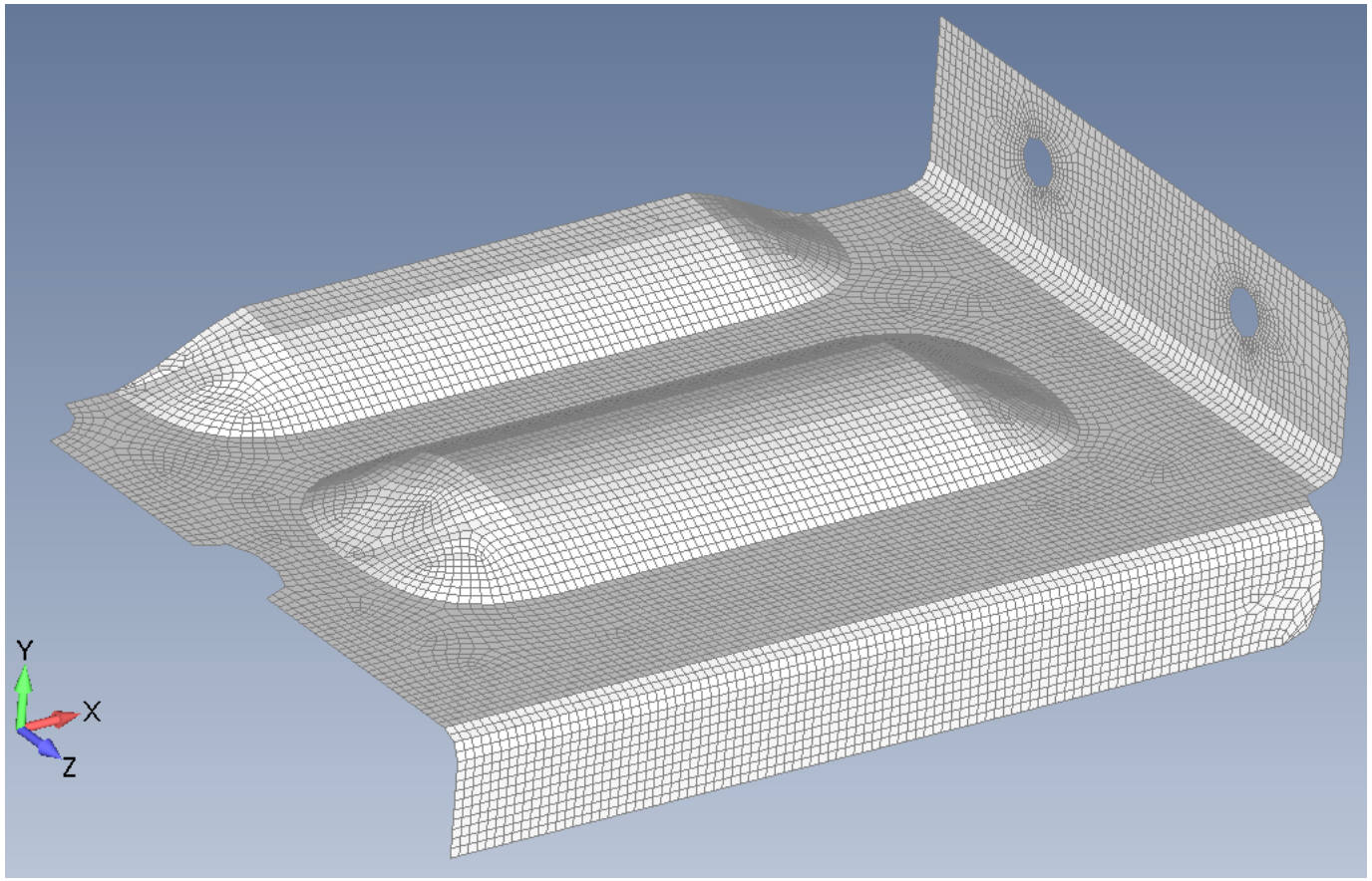
SIEMENS

Body Mesher Technology
Mesh on Mesh Example

FEMComp Engineering
Focusing On FEA

Mesh On Mesh – refined Mesh Sizing for whole model



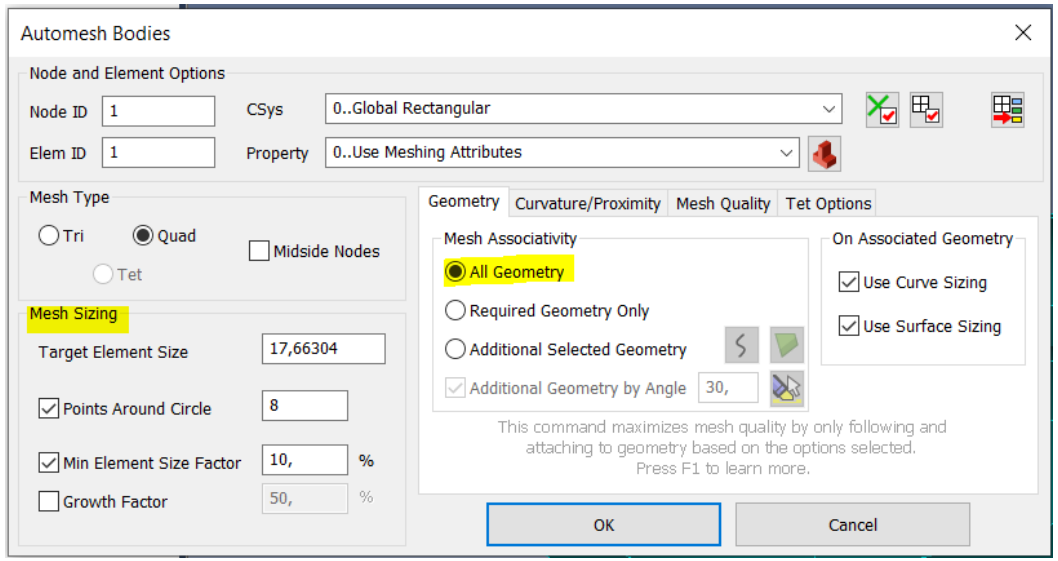


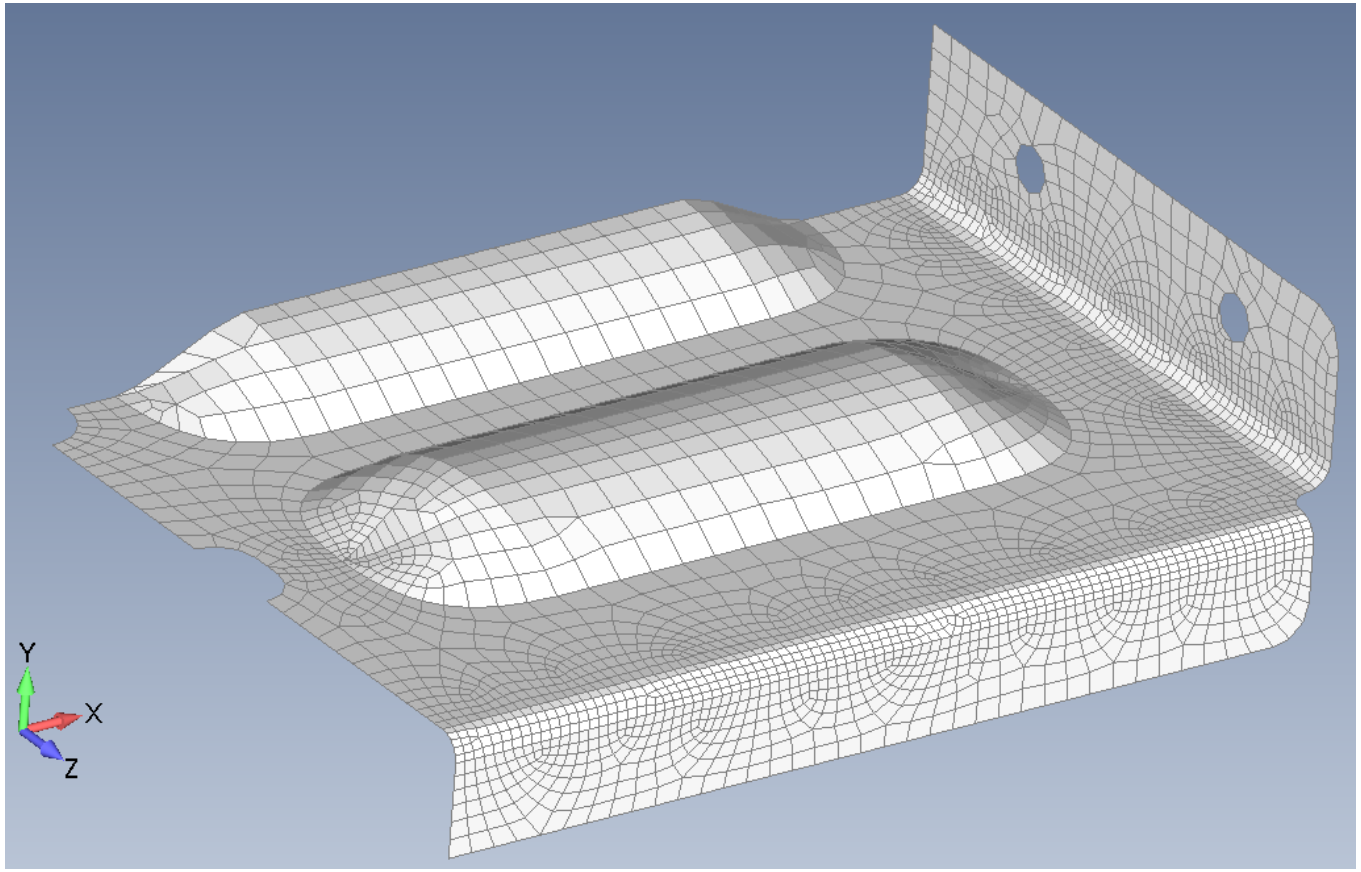
SIEMENS

Body Mesher Technology
Mesh Bodies and Mesh on Mesh Example

FEMComp Engineering
Focusing On FEA

Mesh Bodies – initial Mesh Sizing for whole model



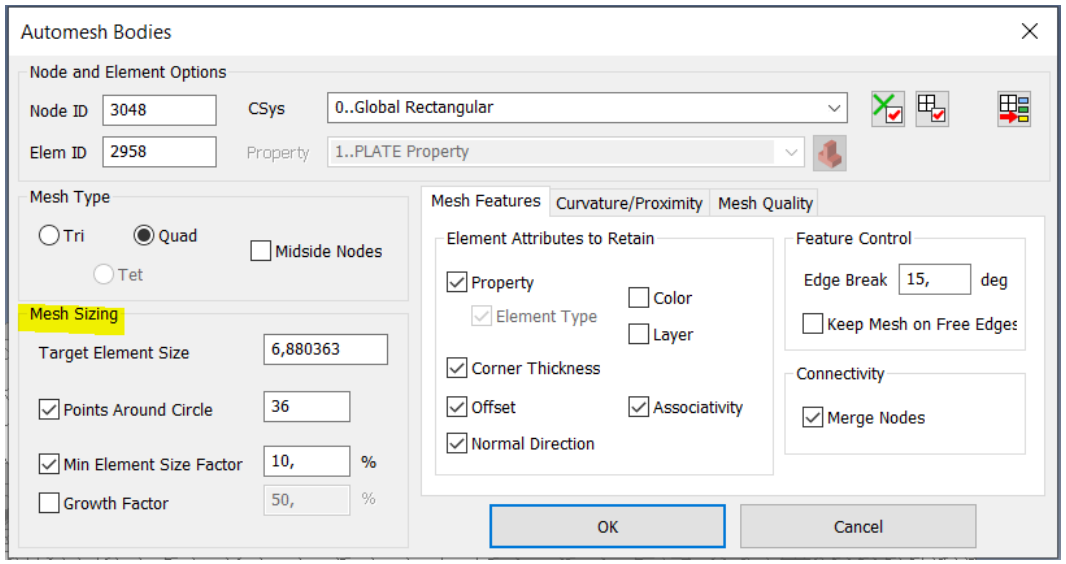


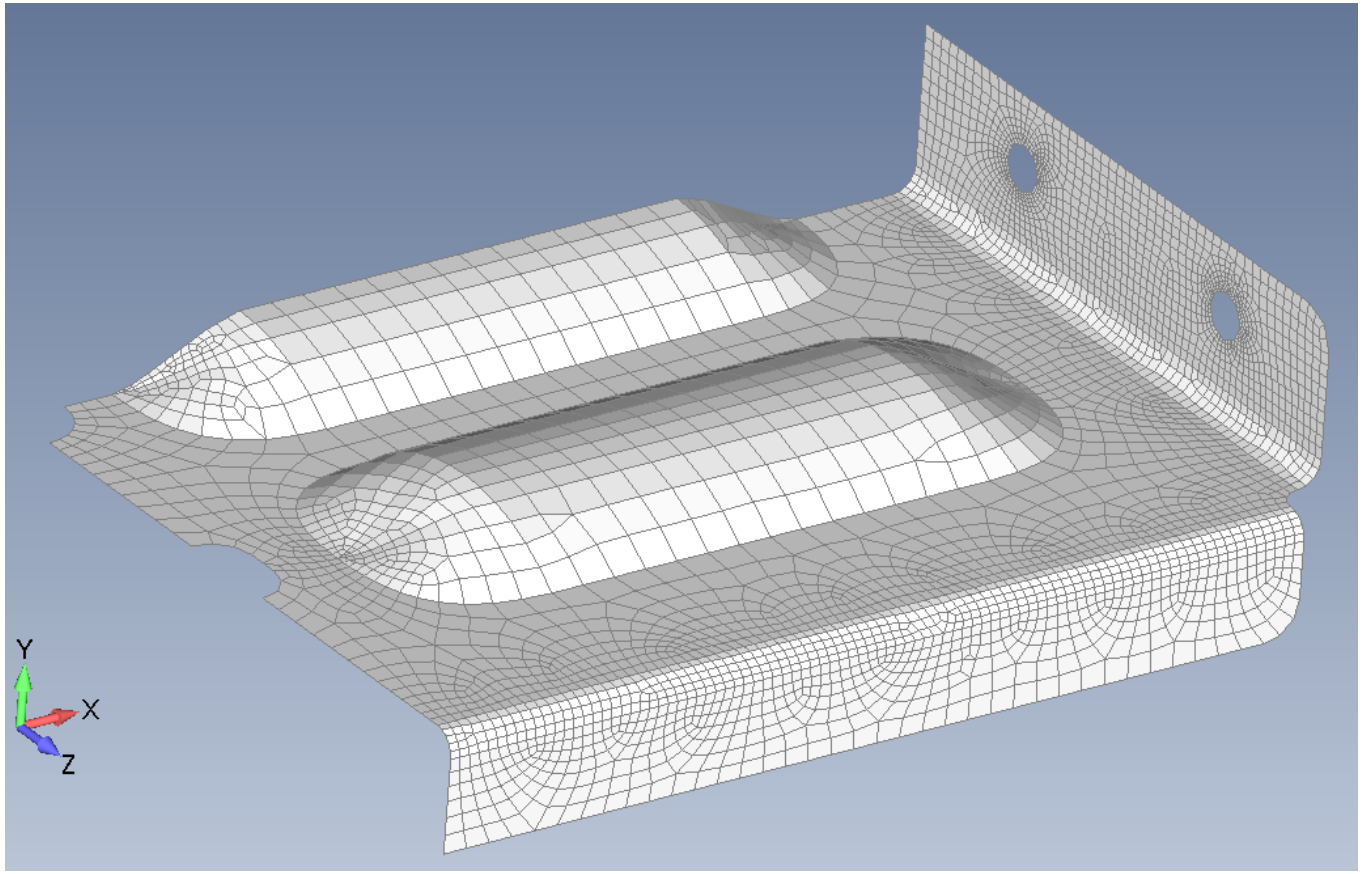
SIEMENS

Body Mesher Technology
Mesh Bodies and Mesh on Mesh Example

FEMComp Engineering
Focusing On FEA

Mesh on Mesh – refined Mesh Sizing specified for **selected** sections

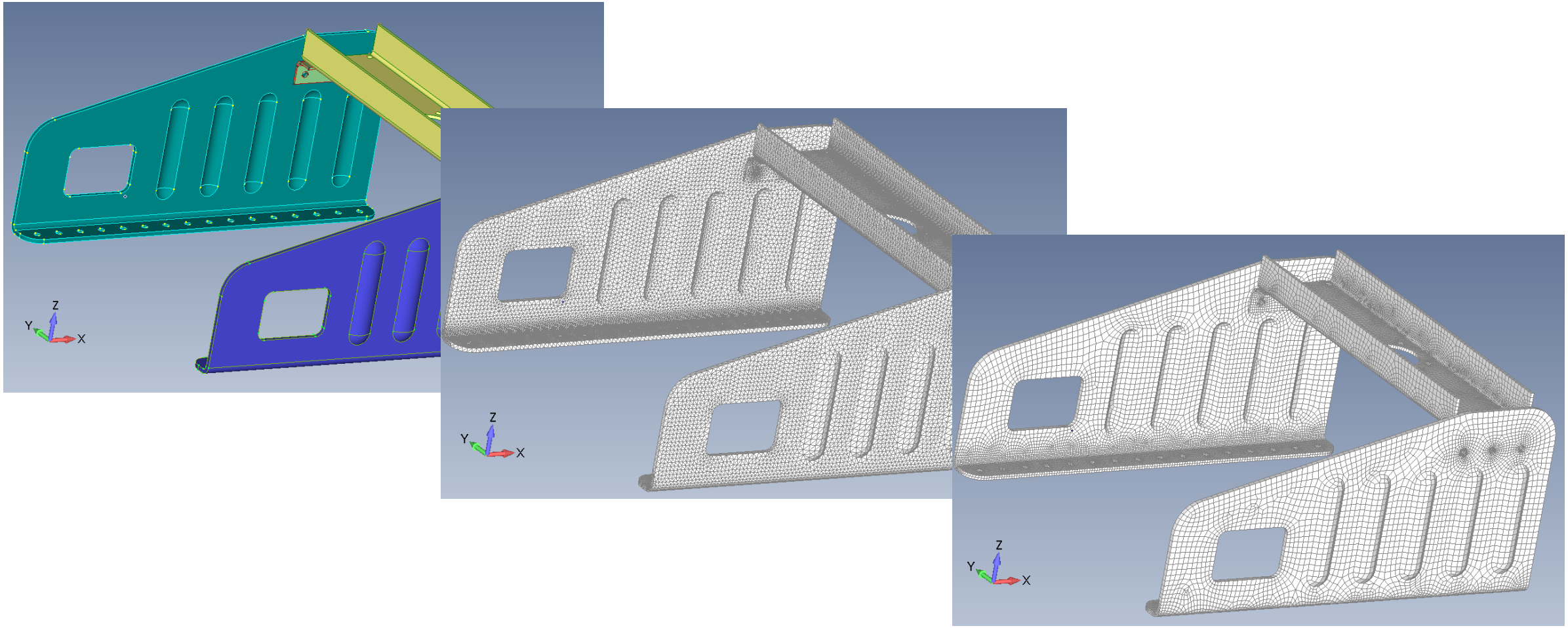




SIEMENS

Body Mesher Technology
Mesh Bodies Solid Assembly Example

Mesh Bodies – initial Mesh Sizing, Tetra and Tetra/Pyramid Solid Mesh



FEMComp Engineering
Focusing On FEA

Simcenter Femap v.2021.2
Body Mesher Technology
Thank you!

SIEMENS