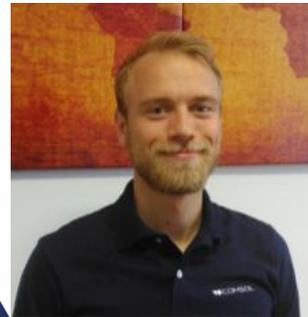
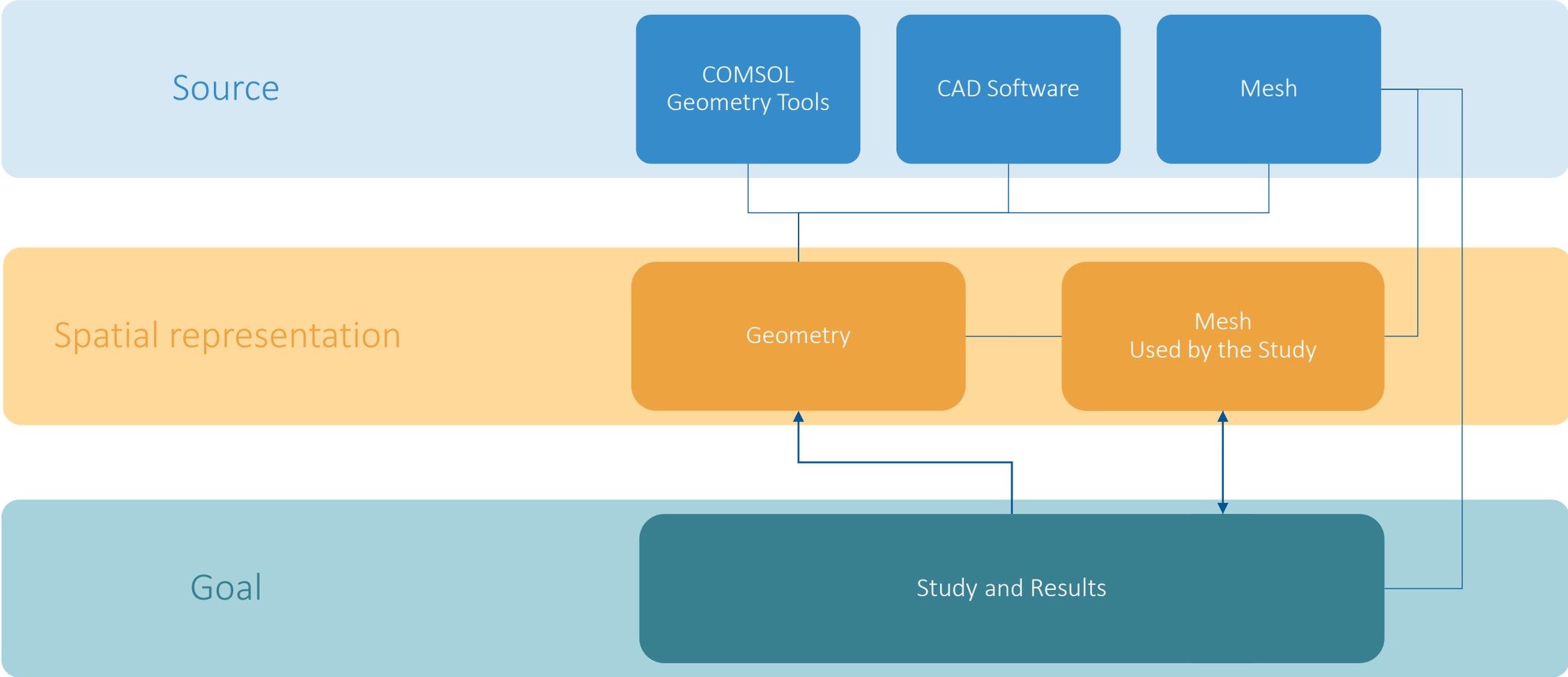


Import and Editing of Segmented Geometries in COMSOL Multiphysics



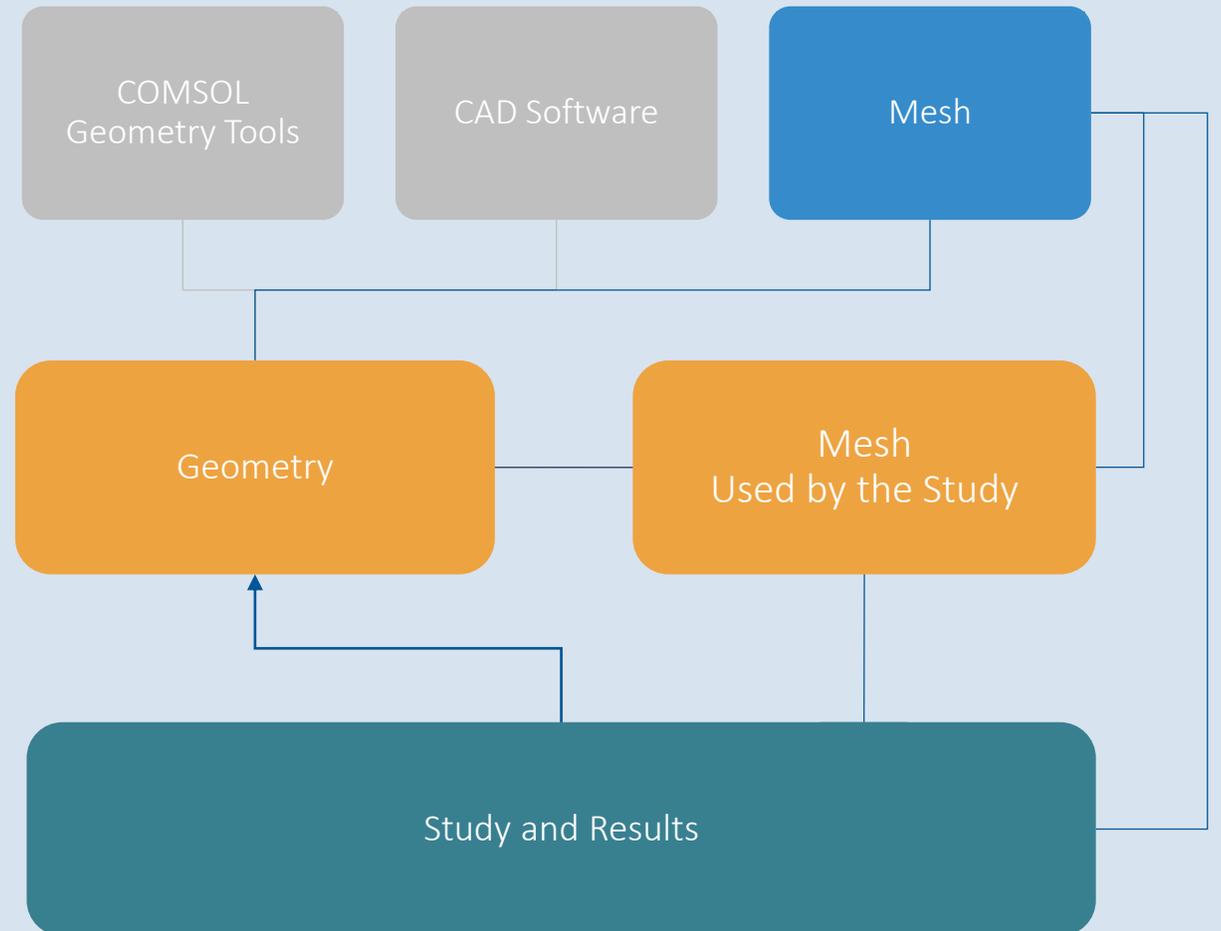
Matouš Lorenc
lorenc@humusoft.cz

The Big Picture



Working With Meshes

- Meshing Concepts
- Preparing the Geometry
- Meshing the Geometry
- Generating Meshes
- Mesh During Solving
- Editing Meshes



The COMSOL[®] Mesh Format

The native file format supports grouping elements into selections and gives full control over how the elements form domains and boundaries.

For more information, see the [COMSOL Multiphysics Mesh Import and Export Guide](#).

Bracket — Topology Optimization - COMSOL Multiphysics

File Home Definitions Geometry Materials Physics Mesh Study Results Developer

Build Mesh Add Import Export Create Geometry Edit Reset Normal Sizing Free Swept Boundary Boundary Normal Distribution Size More Modify Copy Booleans and Cleanup and Create Delete Join Mesh 3 Mesh 3 Mesh 3 Import/Export Physics Controlled Generators Attributes Operations

Model Builder

- Bracket — Topology Optimization
 - Global Definitions
 - Geometrical Parameters
 - Parameters 2
 - Default Model Inputs
 - Materials
 - Load and Constraint Groups
 - Component 2
 - Definitions
 - Geometry 2
 - Materials
 - Solid Mechanics
 - Linear Elastic Material 1
 - Free 1
 - Initial Values 1
 - Fixed Constraint 1
 - Fixed Constraint 2
 - Fixed Constraint 3
 - Fixed Constraint 4
 - Boundary Load 1
 - Boundary Load 2
 - Mesh 3
 - Import 1
 - Finalize
 - Verification
 - Step 1: Stationary
 - Solver Configurations
 - Results
 - Datasets
 - Views
 - Derived Values
 - Tables
 - Displacement (solid2)
 - Surface 1
 - Deformation 1
 - Export
 - Reports

Settings

Import

Build Selected Build All

Label: Import 1

Import

Source:

COMSOL Multiphysics file

Filename:

C:\COMSOL\optimized_bracket.mphbin

Browse Import

Import selections

Import as linear elements

Import domain elements

Boundary partitioning:

Minimal

Domain Selections

Boundary Selections

Name	Name in file
Constraint 1	Constraint 1
Constraint 2	Constraint 2
Constraint 3	Constraint 3
Constraint 4	Constraint 4
Load 1a	Load 1a
Load 1b	Load 1b
Load 2a	Load 2a
Load 2b	Load 2b
Load 1	Load 1
Load 2	Load 2
Material Boundaries	Material Boundaries

Information

Last build time: < 1 second

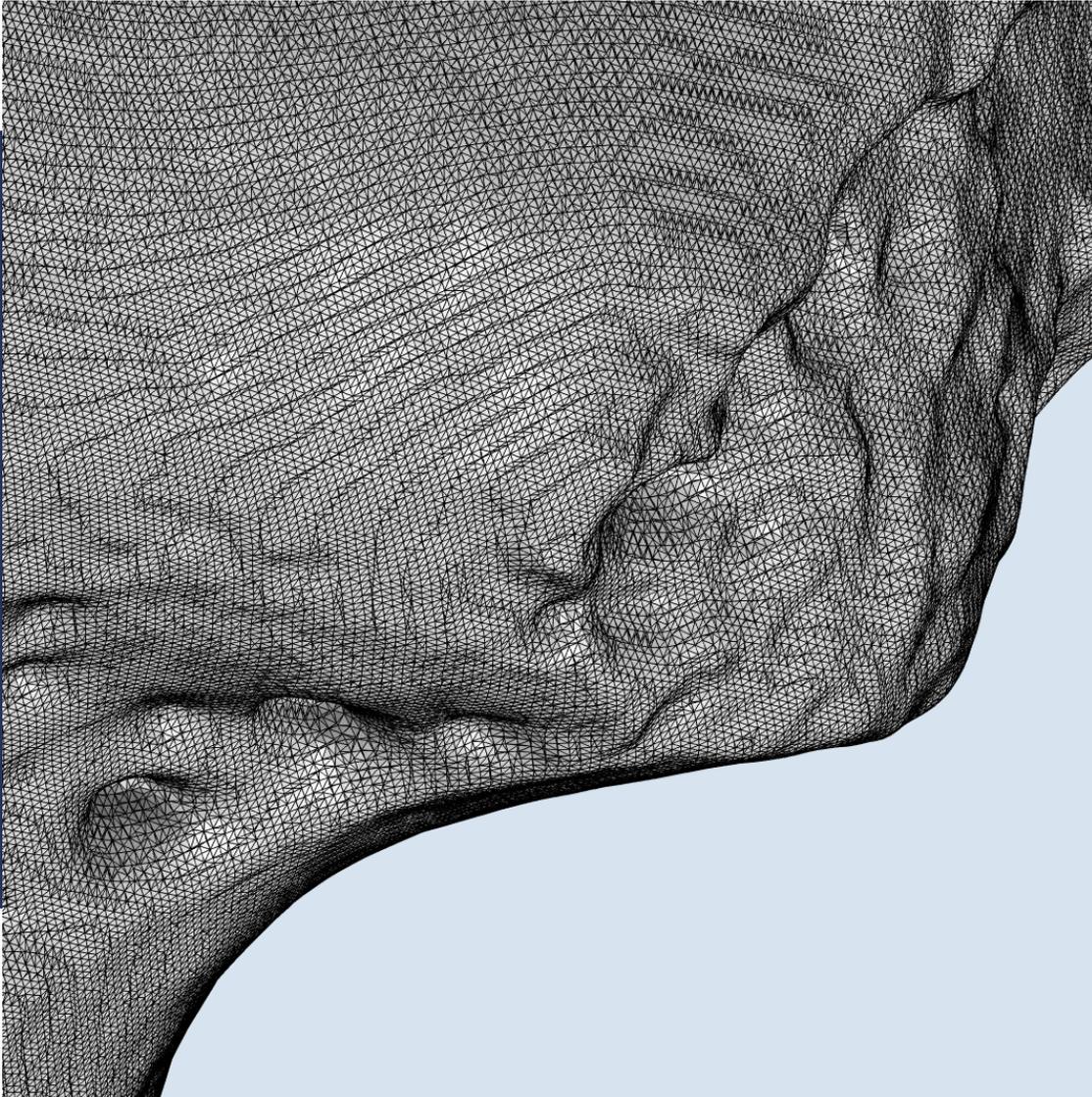
Built with: COMSOL 6.1.0.252 (win64), Nov 28, 2022, 9:36:06

Graphics

Messages Progress Log Objective Probe Table 1

COMSOL Multiphysics 6.1.0.252

798 MB | 2125 MB



STL, PLY, and 3MF Import

Import surface meshes of scanned data to use for simulations. The triangles in the file define a linear mesh, but the software will automatically curve the elements when needed.

"Human Femur". (<https://skfb.ly/6ursh>) by Eric Bauer is licensed under the Creative Commons Attribution license (<http://creativecommons.org/licenses/by/4.0/>).

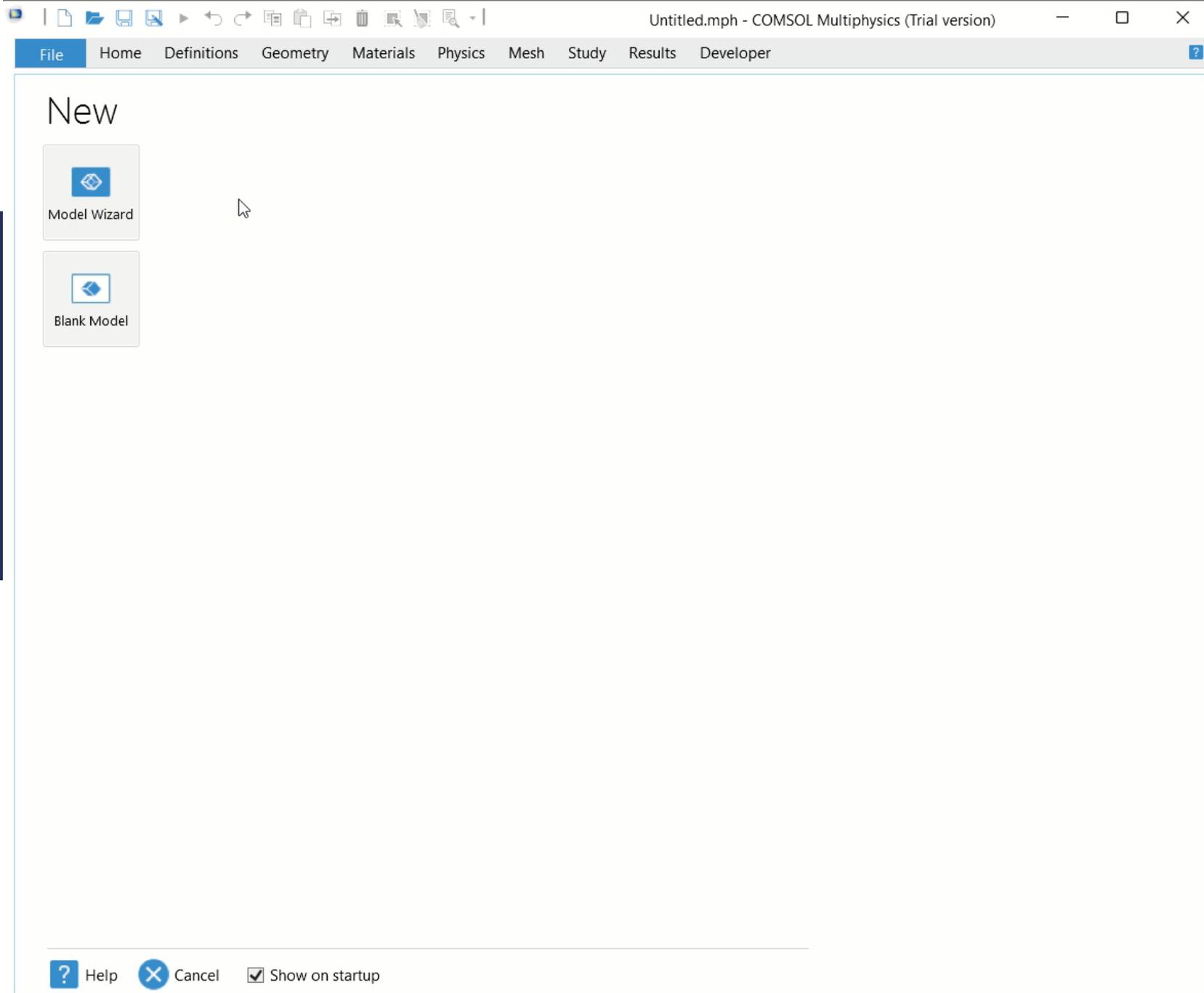
DEMO: Lungs Import

- Segmented geometry of lungs generated by Medical Imaging Toolbox in MATLAB and stored as STL file
- Remeshing and repair of imported mesh
- Combining imported mesh with local geometry
- Visualization and statistic methods



Lungs Import: Importing STL

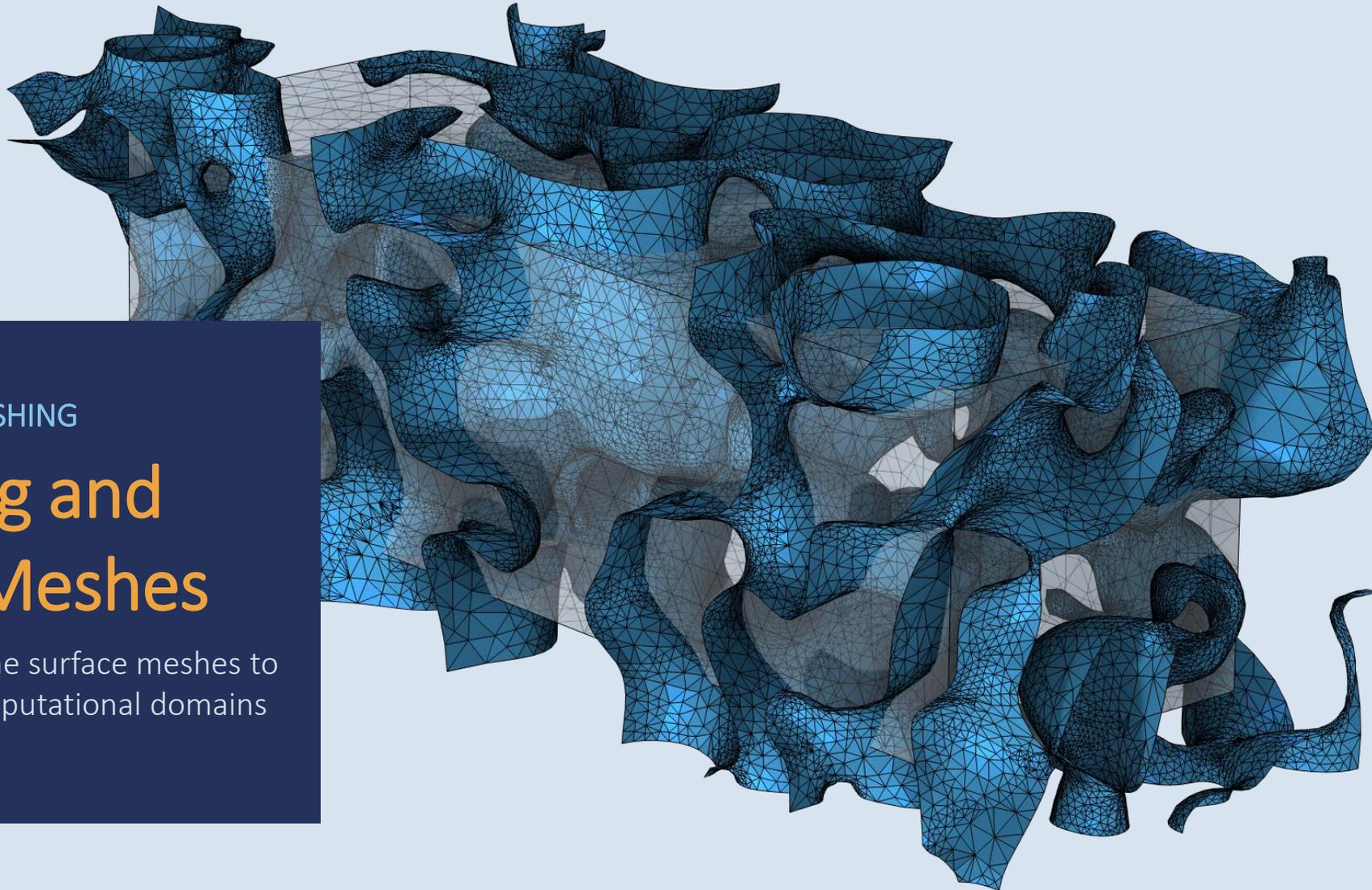
- Minimization of imported boundaries improves the imported mesh quality
- Visible effect of segmentation to be further smoothed out

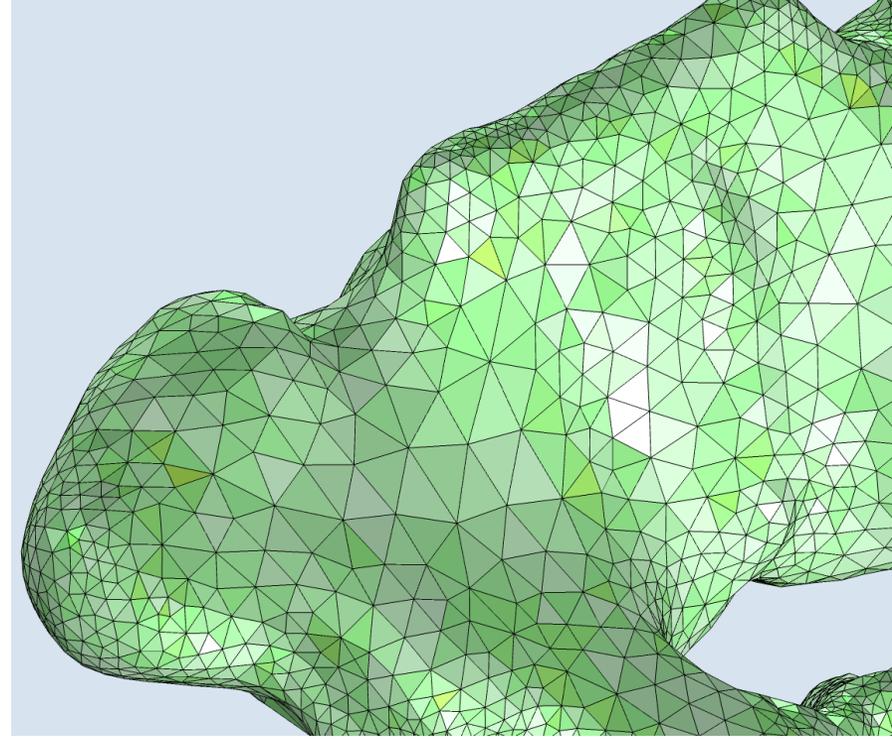
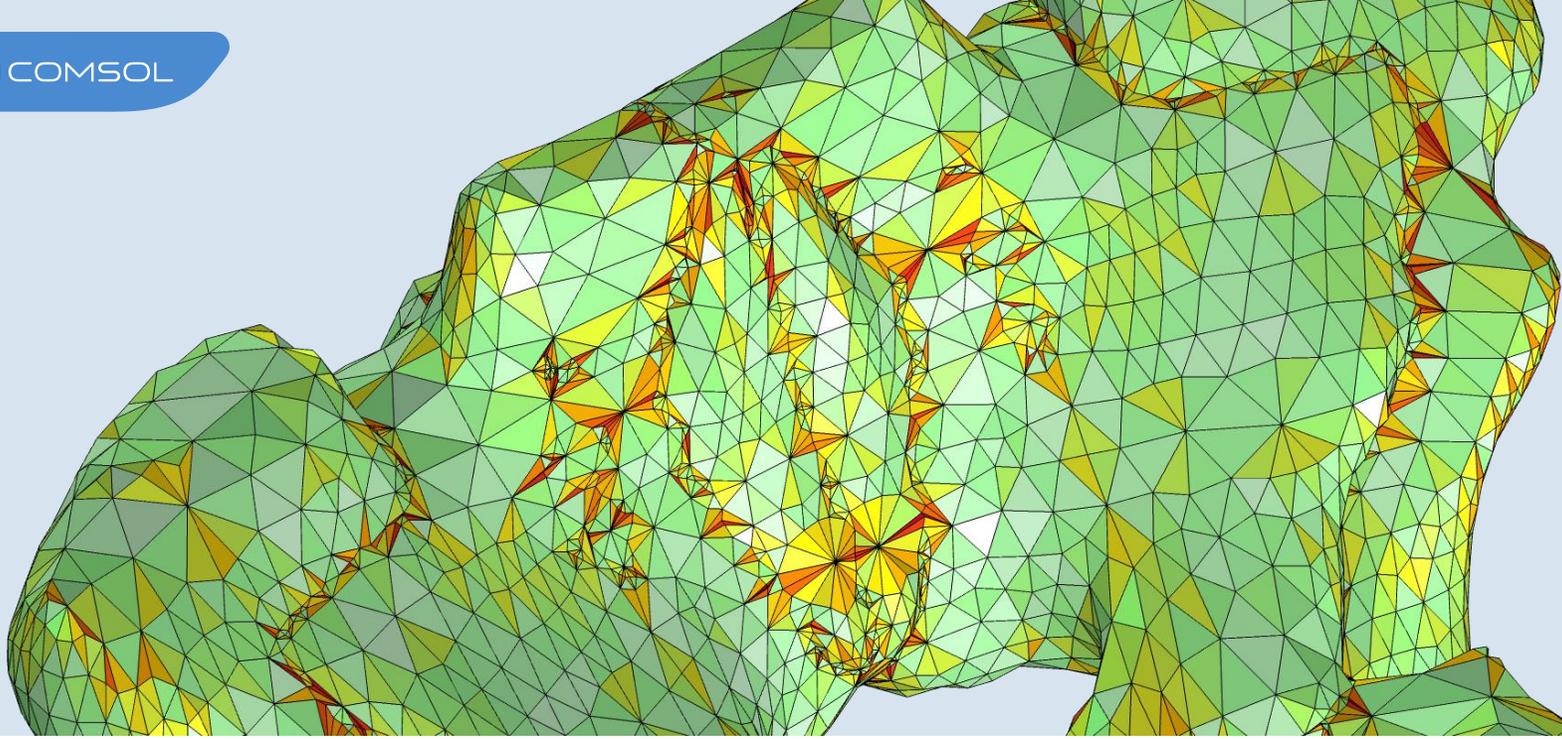


OVERVIEW OF MESHING

Repairing and Editing Meshes

Repair and combine surface meshes to generate new computational domains for simulations.





Modifying Imported Meshes

- Generate new mesh or modify the surface mesh to improve quality and control the element size.
- Watertight regions in surface meshes can be filled with tetrahedral, swept, and boundary layer mesh.

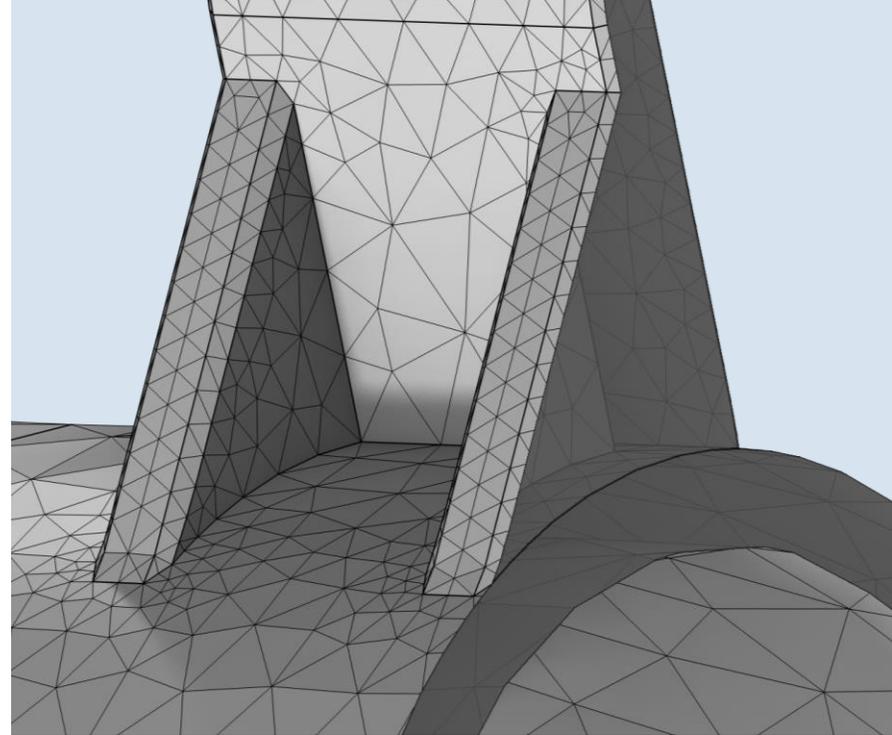
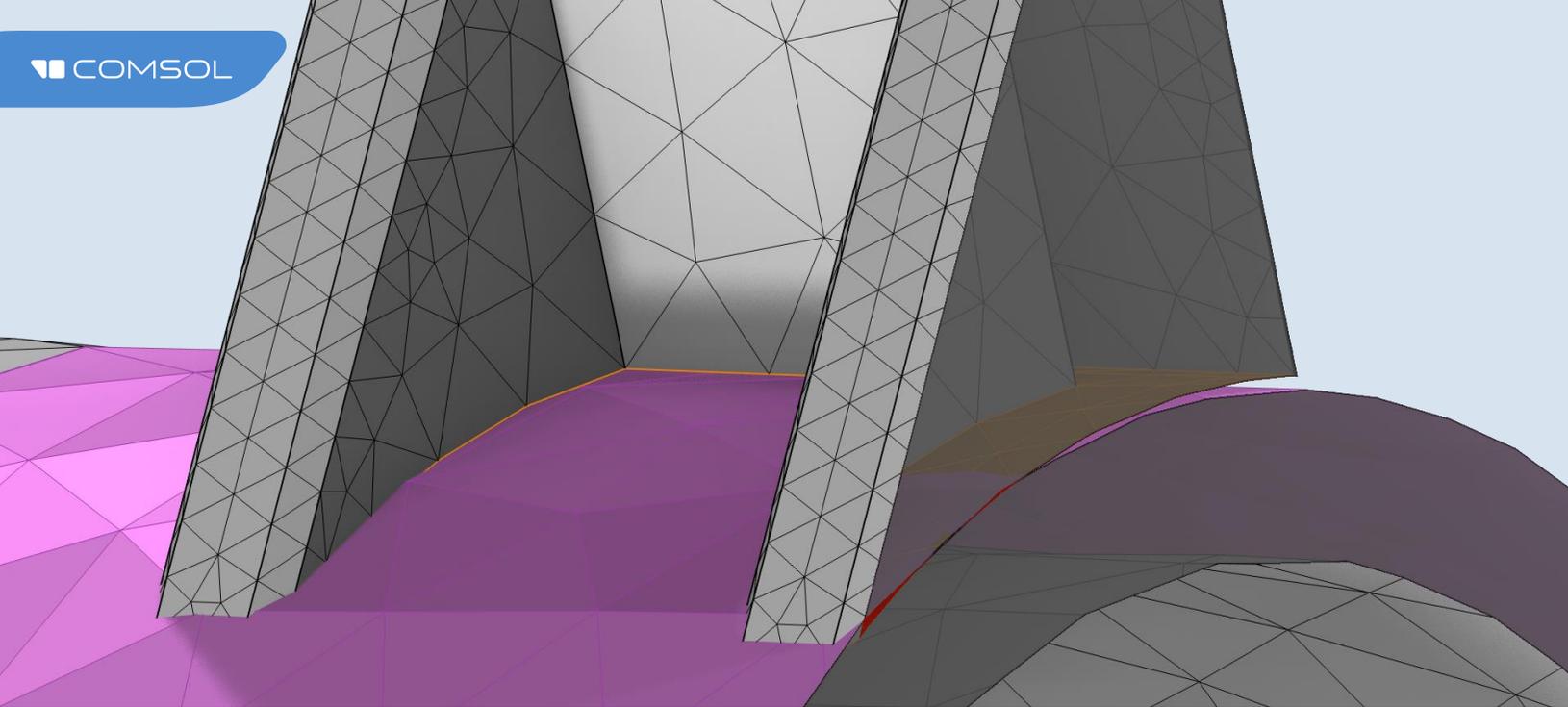
Lungs Import: Remeshing Surface Mesh

- Remeshing surface mesh with *Free Triangular* operation with minimal size value identical to the segmentation value – 0.5 cm

The screenshot displays the COMSOL Multiphysics interface for a model named 'Untitled.mph'. The 'Model Builder' tree on the left shows the hierarchy: Component 1 (comp1) > Mesh 1 > Import 1 > Delete Entities 1. The 'Settings' window for 'Delete Entities 1' is open, showing the 'Geometric Entity Selection' section with 'Geometric entity level' set to 'Boundary' and 'Selection' set to 'Manual'. A list of 8 boundary entities is visible. The 'Adjacent Entities' section has the checkbox 'Delete adjacent lower-dimensional entities' checked. The 'Information' section shows the last build time as '< 1 second' and the version as 'COMSOL 6.1.0.357 (win64), May 23, 2023'. The 'Graphics' window on the right shows a 3D view of the lung mesh with axes labeled in mm (0 to 200). The 'Messages' window at the bottom shows the following log entries:

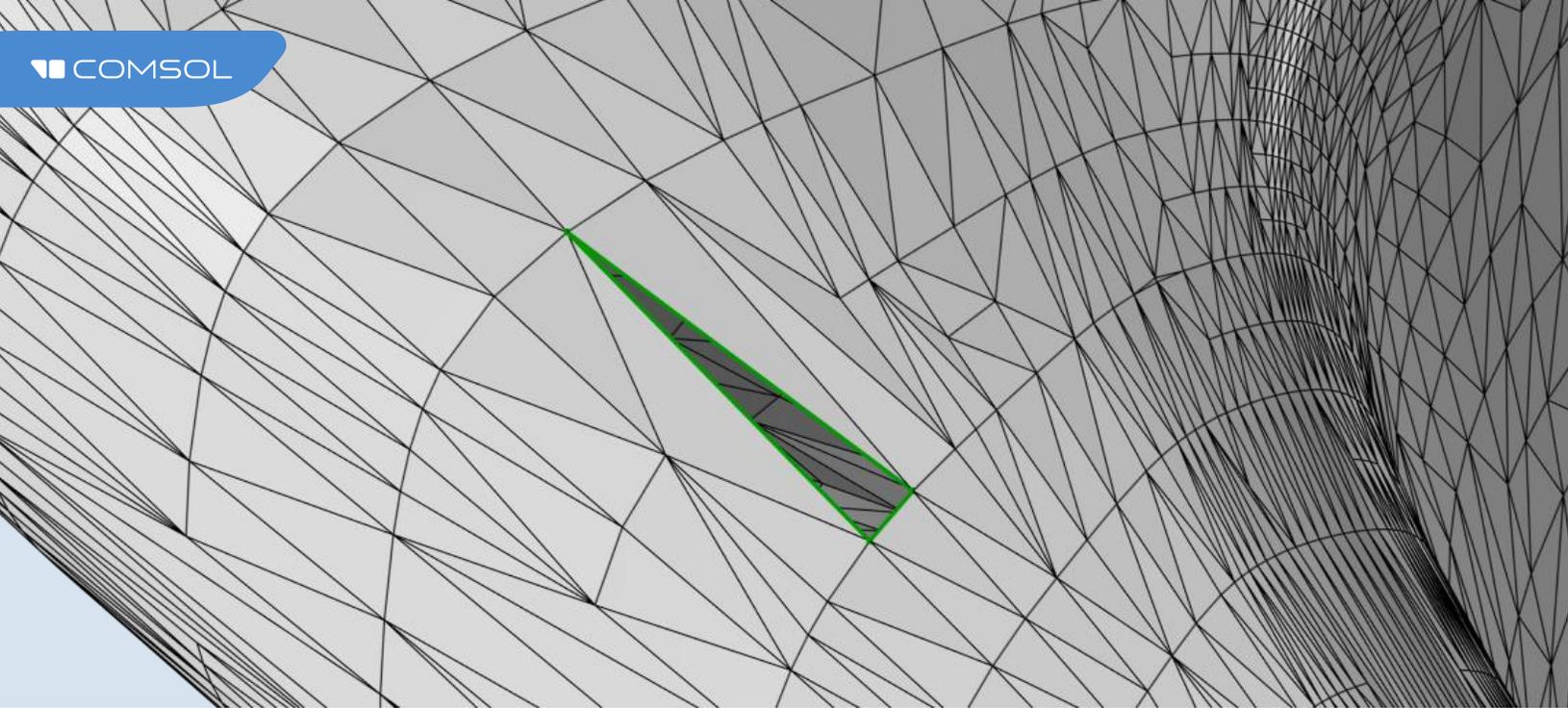
```

[May 23, 2023, 10:30 PM] Finalized geometry is empty.
[May 23, 2023, 10:30 PM] Imported mesh with 333024 boundary eleme
[May 23, 2023, 10:31 PM] Imported mesh with 333024 boundary eleme
[May 23, 2023, 10:31 PM] Mesh has 8 boundaries (333024 elements), 1C
[May 23, 2023, 10:31 PM] Mesh has 2 boundaries (332788 elements).
  
```



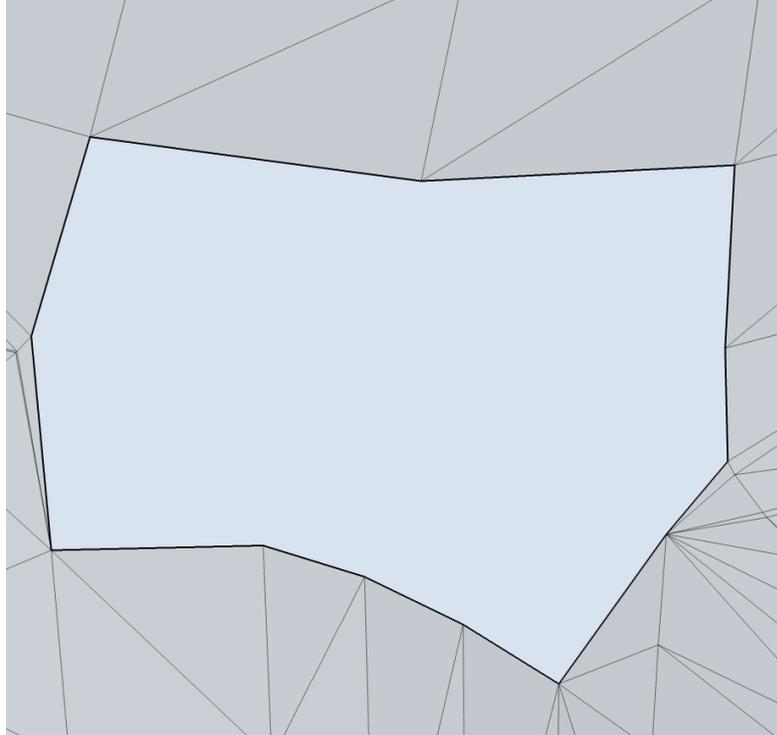
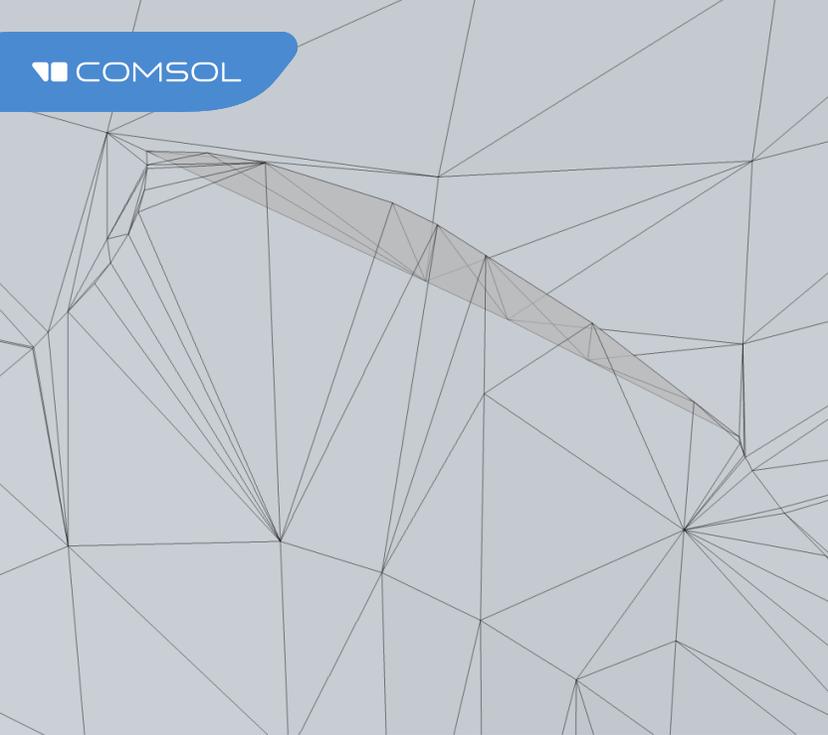
Repairing Small Overlaps and Gaps

- Use to repair imported surface meshes or as an alternative to geometry repair for misaligned CAD models
- Merge of nearby faces, edges, or vertices in meshes for easier meshing



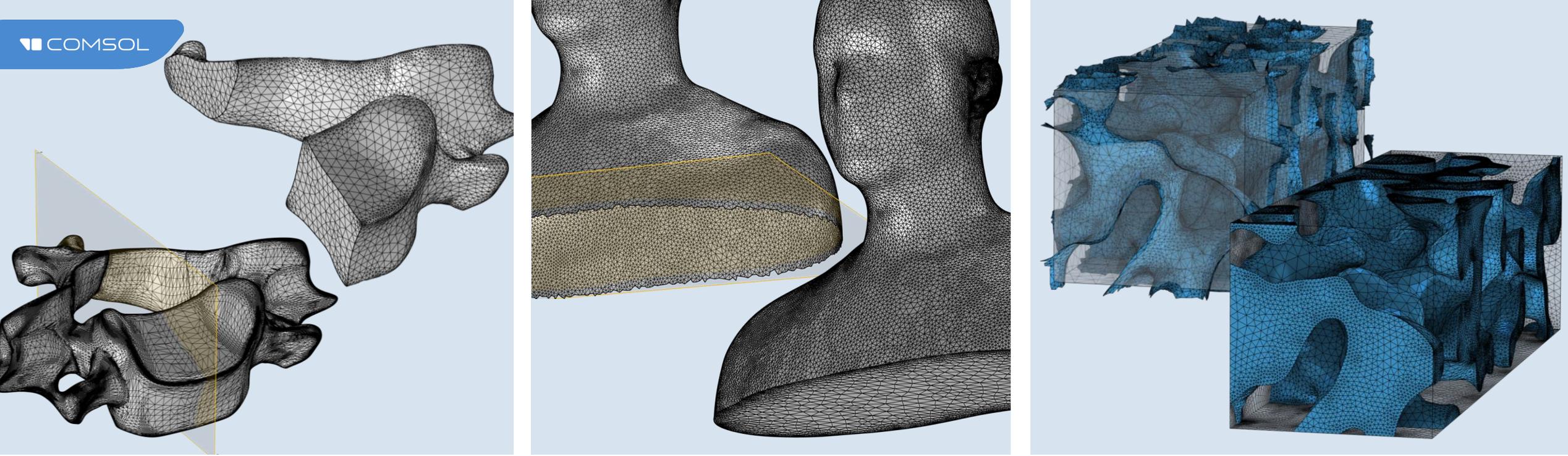
Repairing Holes

- Automatic repair of small holes during mesh import
- Use the *Fill Holes* operation to automatically repair holes based on hole perimeters
- Use *Create Faces* to generate meshed faces after manually selecting hole edges



Removing Sharp Irregularities

- Automatic removal of smaller irregularities when:
 - Creating geometry from mesh
 - Remeshing faces
- Manually isolate irregularities and replace with new meshed faces



Creating Domains from Surface Meshes

- Unite meshes to form new computational domains
 - Automatic cleanup of small elements
- Intersect meshes with a plane (3D) or straight line (2D)
 - Intersection faces created within edge loops

Lungs Import: Repairing Imported Mesh

- Surface mesh needs to be repaired to create volume
- Faulty elements can cut out and replaced by new mesh
- Only now domains can be formed

Untitled.mph - COMSOL Multiphysics (Trial version)

File Home Definitions Geometry Materials Physics Mesh Study Results Developer

Application Builder Component 1
Model Manager Add Component

Parameters Variables
a= Variables
foo
P1

Geometry Materials
Select Physics Interface
Add Physics

Mesh Study Results Layout

Workspace Model Definitions Physics

Model Builder

- Untitled.mph (root)
 - Global Definitions
 - Parameters 1
 - Materials
 - Component 1 (comp1)
 - Definitions
 - Geometry 1
 - Materials
 - Mesh 1
 - Import 1
 - Information
 - Delete Entities 1
 - Free Triangular 1
 - Size
 - Finalize
 - Results

Settings

Size

Build Selected Build All

Label: Size

Element Size

Calibrate for:
General physics

Predefined Finer

Custom

Element Size Parameters

Maximum element size:
10 mm

Minimum element size:
5 mm

Maximum element growth rate:
1.2

Curvature factor:
0.4

Resolution of narrow regions:
0.7

Graphics

mm

150

250

200

150

100

50

200

150

100

50

mm

z

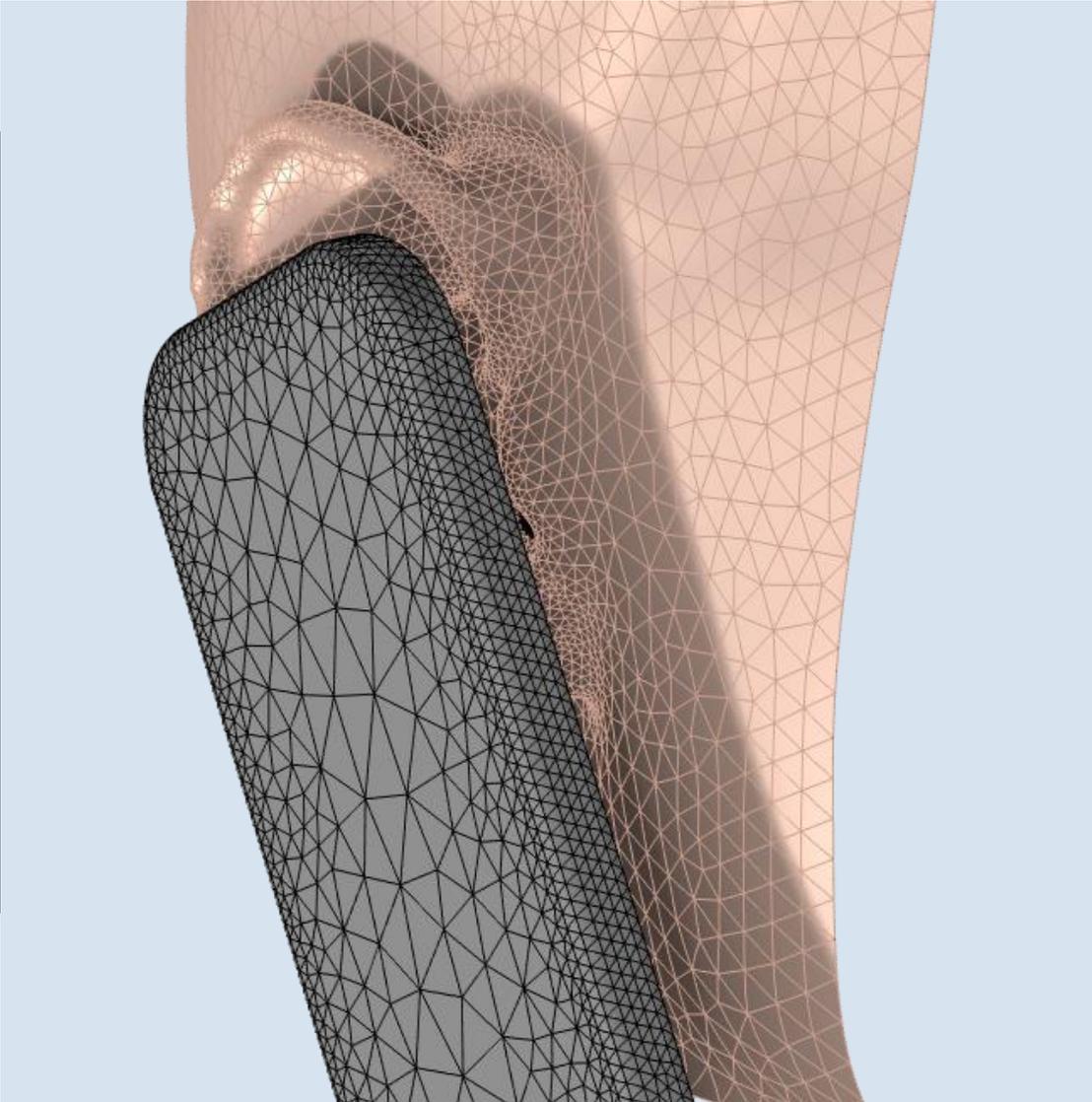
y x

Messages

Progress Log Table

[May 23, 2023, 10:39 PM] Mesh error (dom 1): intersecting face element
 [May 23, 2023, 10:40 PM] Mesh has 3 boundaries (14514 elements), 1 e
 [May 23, 2023, 10:40 PM] Mesh has 2 boundaries (14459 elements), 1 e

1.05 GB | 1.17 GB



Combining Imported Meshes with CAD

The geometry of a cellphone is united with the scanned data of an ear.

The geometry can be created in COMSOL Multiphysics® or imported from CAD files and then be combined with the imported mesh.

Lungs Import: Joining with Geometry

- Object created under *Geometry* node handled by an imported mesh node
- Each object still has its own mesh unaffected by the other

Untitled.mph - COMSOL Multiphysics (Trial version)

File Home Definitions Geometry Materials Physics Mesh Study Results Developer

Application Builder Component 1
Model Manager Add Component

Parameters Variables
a= Variables
foo
Pi

Geometry Materials

Select Physics Interface
Add Physics

Mesh Study Results Layout

Workspace Model Definitions Physics

Model Builder

Untitled.mph (root)

- Global Definitions
 - Parameters 1
 - Materials
- Component 1 (comp1)
 - Definitions
 - Geometry 1
 - Materials
 - Mesh 1
 - Import 1
 - Delete Entities 1
 - Free Triangular 1
 - Create Edges 1
 - Delete Entities 2
 - Fill Holes 1
 - Create Domains 1
 - Finalize
 - Results

Settings

Create Domains

Build Selected Build All

Label: Create Domains 1

Information

Last build time: < 1 second
Built with: COMSOL 6.1.0.357 (win64), May 23, 2023,

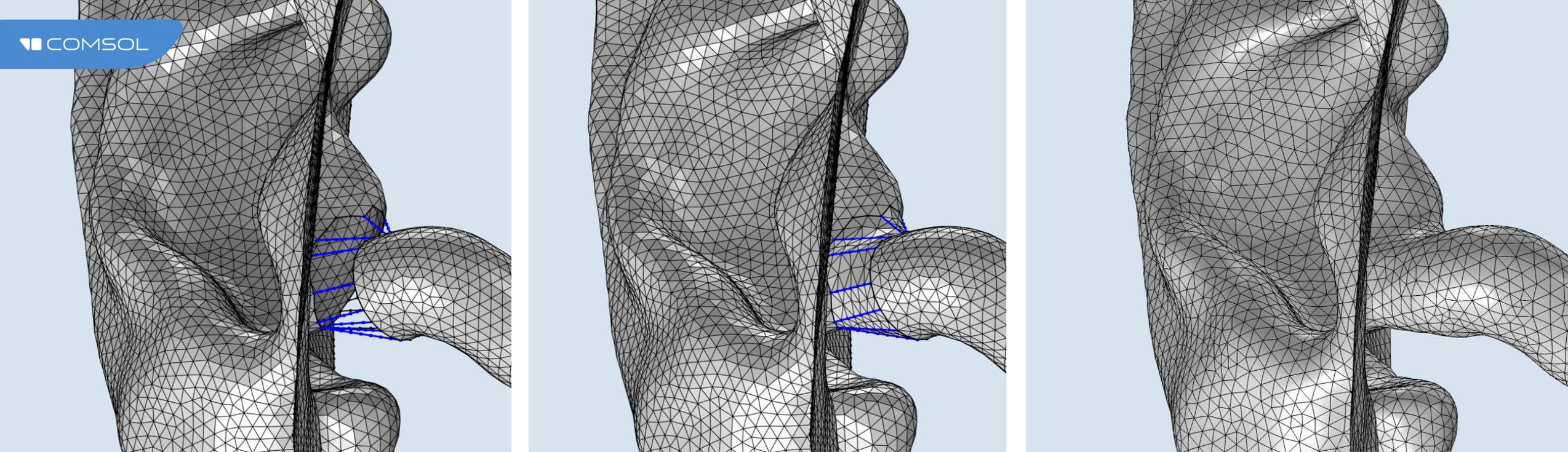
Graphics

Messages

Progress Log Table

[May 23, 2023, 10:43 PM] Mesh has 3 boundaries (14514 elements), 1 e
[May 23, 2023, 10:44 PM] Mesh has 2 boundaries (14474 elements), 1 e
[May 23, 2023, 10:44 PM] Mesh has 1 domain and 2 boundaries (14494
[May 23, 2023, 10:44 PM] Mesh has 2 domains and 2 boundaries (14494

1.16 GB | 1.28 GB



Connecting Meshes

Bridge gaps between meshes by creating meshed edges and faces, and then generate new mesh to improve the mesh quality.

Lungs Import: Mesh Finalization

- *Union* operation unifies the mesh of the geometry object together with the lungs and creates one continuous mesh
- Operation *Finalize* checks the mesh compliance with FEM requirements

The screenshot displays the COMSOL Multiphysics interface for an 'Untitled.mph' file. The top toolbar includes menus for File, Home, Definitions, Geometry, Materials, Physics, Mesh, Study, Results, and Developer. The main workspace is divided into three panels: Model Builder, Settings, and Graphics.

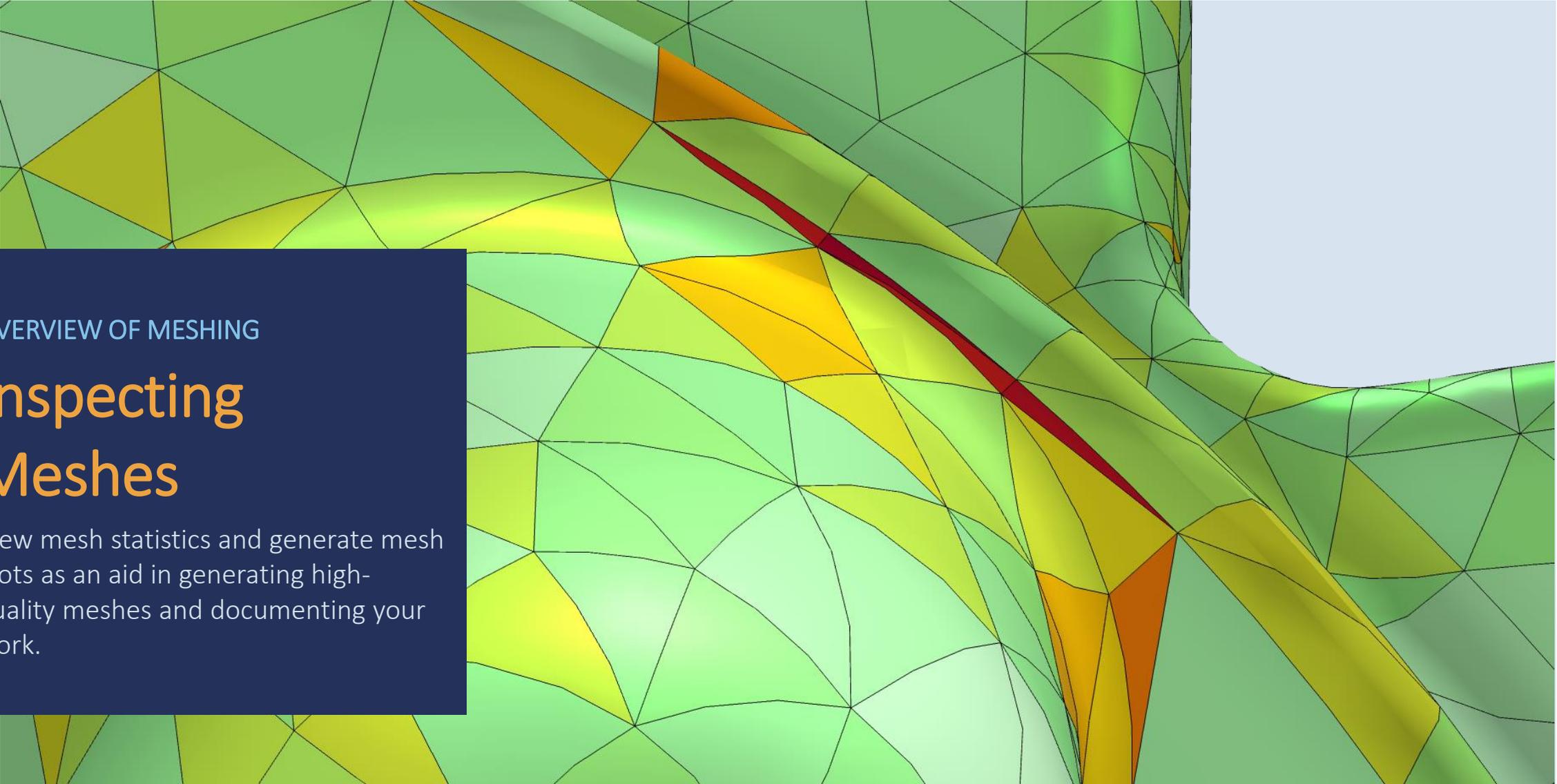
Model Builder: Shows a hierarchical tree of the model. Under 'Component 1 (comp1)', 'Geometry 1' is expanded to show 'Work Plane 1 (wp1)', 'Plane Geometry', and 'Ellipse 1 (e1)'. Below this, 'Extrude 1 (ext1)' is selected, followed by 'Form Union (fn)'. The 'Mesh 1' section is also visible, with 'Import 2' highlighted.

Settings: The 'Import' settings are shown. The 'Label' is 'Import 2'. The 'Source' is set to 'Geometry sequence'. The 'Geometry' is 'Geometry 1'. The 'Face mesh generation' is set to 'Free triangular'. The 'Size' is 'Normal'. The 'Resolve narrow domain regions' checkbox is checked. The 'Information' section shows 'Last build time: < 1 second' and 'Built with: COMSOL 6.1.0.357 (win64), May 23, 2023'.

Graphics: A 3D visualization of the lung model is shown. The model is a complex, curved structure with a fine mesh. The axes are labeled x, y, and z. The dimensions are 200 mm in the x and z directions, and 150 mm in the y direction.

Messages: A log of messages is shown at the bottom. The messages indicate the progress of the meshing process:

- [May 23, 2023, 10:44 PM] Mesh has 1 domain and 2 boundaries (14494
- [May 23, 2023, 10:44 PM] Mesh has 2 domains and 2 boundaries (14494
- [May 23, 2023, 10:49 PM] Imported mesh with 784 boundary elements :
- [May 23, 2023, 10:49 PM] Mesh has 3 domains, 8 boundaries (15278 ele



OVERVIEW OF MESHING

Inspecting Meshes

View mesh statistics and generate mesh plots as an aid in generating high-quality meshes and documenting your work.

Plotting the Mesh

Create high-quality meshes by detecting low-quality elements and making appropriate adjustments.

submarine_cable_07_geom_mesh_3d.mph - COMSOL Multiphysics

File Home Definitions Geometry Materials Physics Mesh Study Results Developer Mesh Quality, Volume Elements

Volume Slice Line Arrow Line Image Color Expression Material Appearance Evaluate Along Normal Cut Line Direction Second Po
 Arrow Volume Isosurface Contour Mesh Deformation Selection More Plots Filter Transparency More Attributes
 Surface Arrow Surface Streamline Annotation Attributes Select

Plot Plot In Add Plot

Model Builder

- submarine_cable_07_geom_mesh_3c
 - Global Definitions
 - Component 1
 - Definitions
 - Geometry 1
 - Materials
 - Mesh 1
 - Size
 - Free Triangular 1
 - Mapped 1
 - Copy Face 1
 - Free Triangular 2
 - Mapped 2
 - Copy Face 2
 - Mapped 3
 - Free Triangular 3
 - Mapped 4
 - Swept 1
 - Convert 1
 - Copy Face 3
 - Free Tetrahedral 1
 - Results
 - Datasets
 - Mesh 1
 - Cut Plane 1
 - Cut Plane 2
 - Views
 - Derived Values
 - Tables
 - Mesh Quality, Volume Eleme
 - Mesh 1
 - Mesh Quality, Poor Quality El
 - Mesh 1
 - Selection 1
 - Mesh 2
 - Selection 1
 - Mesh 3
 - Mesh Comparison, Source ar
 - Surface 1
 - Surface 2
 - Deformation 1
 - Export
 - Reports

Settings

Mesh

Plot

Label: Mesh 1

Data

Dataset: From parent

Title

Title type: Automatic

Level

Level: Volume

Element type: All

Coloring and Style

Element color: Quality

Quality measure: Skewness

Color range [0,1]

Color table: TrafficFlow

Color legend

Color table transformation: Nonlinear

Reverse color table

Color calibration parameter: 0

Color table type: Continuous

Wireframe color: Black

Element Filter

Enable filter

Criterion: Logical expression

Expression: $z < Lsec / 2 - 2 * Nper * (Dcab + x + y)$

Shrink Elements

Inherit Style

Graphics

Mesh: Skewness

0.949

1

0.9

0.8

0.7

0.6

0.5

0.4

0.3

0.2

0.1

0

0.0281

Messages Progress Log Evaluation 3D

COMSOL Multiphysics 6.1.0.252

1.36 GB | 3.15 GB

Mesh Statistics

View the element quality histogram and other statistics, such as the minimum and average element quality.

The screenshot displays the COMSOL Multiphysics interface for a simulation titled "wax_guard_acoustics.mph". The interface is divided into several panels:

- Model Builder:** Shows the hierarchical structure of the model, including Global Definitions, Component 1, Geometry 1, Materials, Thermoviscous Acoustics, Frequency Domain, and Mesh 1.
- Statistics Panel:** Provides detailed information about the mesh.

Complete mesh	
Mesh vertices:	10739
Element type:	All elements
Tetrahedra:	11775
Pyramids:	1373
Prisms:	9025
Hexahedra:	2442
Triangles:	3598
Quads:	1258
Edge elements:	741
Vertex elements:	100
Domain element statistics	
Number of elements:	24615
Minimum element quality:	0.06033
Average element quality:	0.6443
Element volume ratio:	3.438E-4
Mesh volume:	1.953 mm ³
- Element Quality Histogram:** A bar chart showing the distribution of element quality across the mesh. The x-axis represents the quality value, and the y-axis represents the number of elements. The histogram shows a peak in the middle range, indicating a good overall mesh quality.
- 3D Mesh Visualization:** A 3D view of the wax guard model with a mesh overlay. A color gradient is applied to the mesh, likely representing the element quality or a physical property like acoustic velocity.

Lungs Import: Mesh Visualization and Statistics

- Creating *Mesh Plot* to check for elements quality and visualization
- Statistical evaluation of Mesh to quantify the overall mesh quality

Untitled.mph - COMSOL Multiphysics (Trial version)

File Home Definitions Geometry Materials Physics Mesh Study Results Developer

Application Builder Component 1 Variables
Model Manager Add Component Parameters
Geometry Materials
Mesh Study Results Layout

Workspace Model Definitions Physics

Model Builder

- Untitled.mph (root)
 - Global Definitions
 - Parameters 1
 - Materials
 - Component 1 (comp1)
 - Definitions
 - Geometry 1
 - Work Plane 1 (wp1)
 - Plane Geometry
 - Ellipse 1 (e1)
 - View 2
 - Extrude 1 (ext1)
 - Form Union (fn)
 - Materials
 - Mesh 1
 - Import 1
 - Delete Entities 1
 - Free Triangular 1
 - Create Edges 1
 - Delete Entities 2
 - Fill Holes 1
 - Create Domains 1
 - Import 2
 - Union 1
 - Free Tetrahedral 1
 - Finalize
 - Results

Settings

Finalize

Build Selected Build All

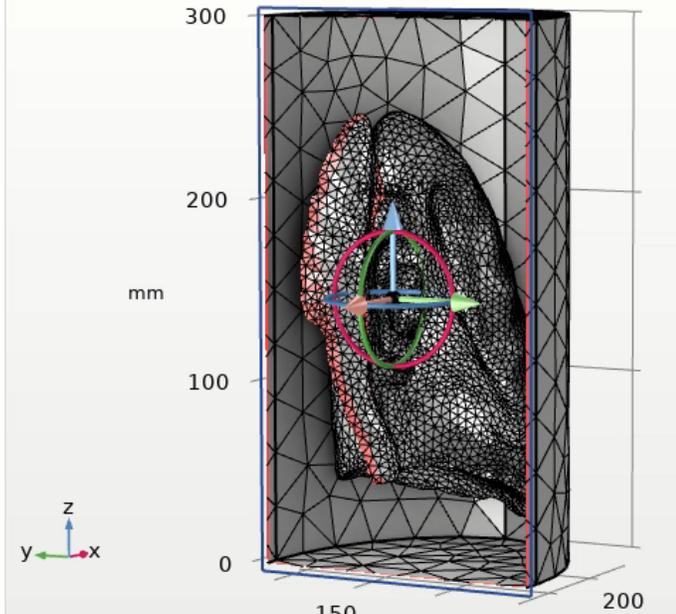
Label: Finalize

Information

Last build time: < 1 second

Built with: COMSOL 6.1.0.357 (win64), May 23, 2023,

Graphics



Messages Progress Log Table

[May 23, 2023, 11:05 PM] Mesh has 3 domains (151932 elements), 8 DO
 [May 23, 2023, 11:09 PM] Finalized mesh has 3 domains (151932 elements)
 [May 23, 2023, 11:15 PM] Finalized mesh has 3 domains (149353 elements)

1.79 GB | 1.94 GB