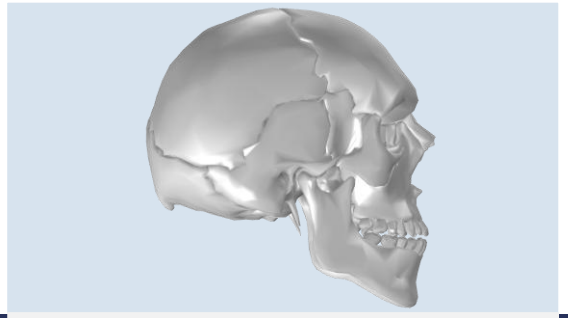
COMSOL Minicourse: Meshing, Biomedical

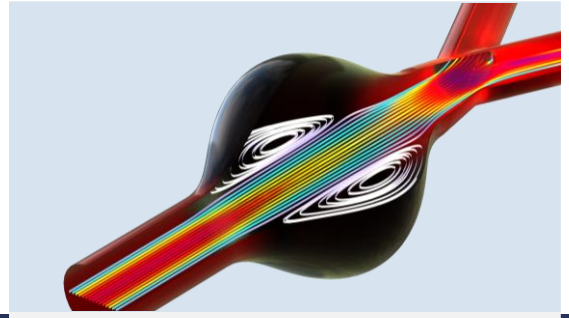
Tomáš Vrbata
vrbata@humusoft.cz

Biomedical Applications of Mesh-Based Geometry



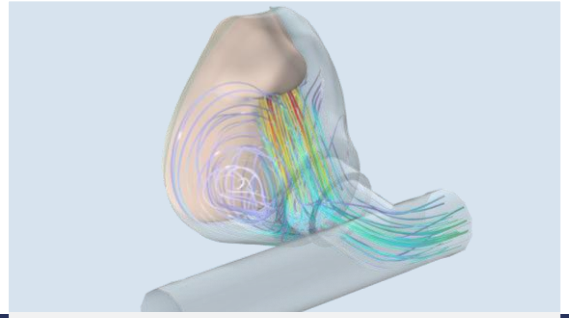
Patient-specific orthopedics

- Bone biomechanics, spine, joints
- Implant fit and stress analysis



Cardiovascular flow

- Blood flow in arteries and aneurysms
- Stent and graft design



Respiratory and drug delivery

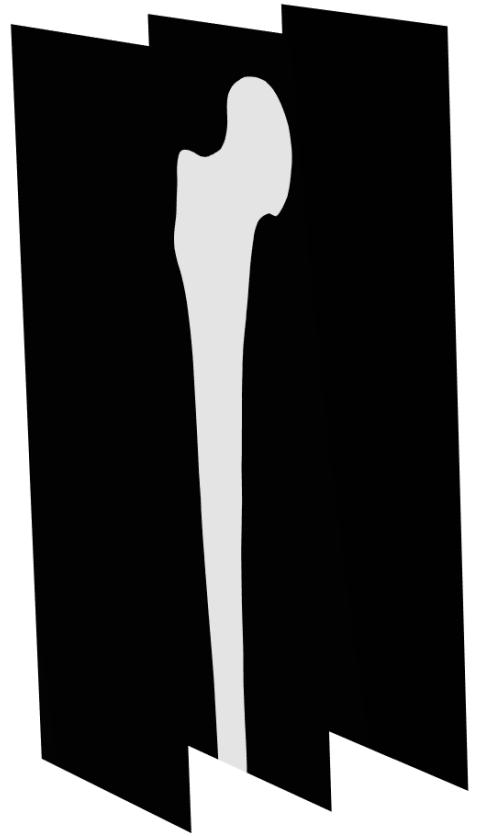
- Airway flow, particle deposition
- Nasal spray, inhaler optimization

Soft tissue

- Tumor ablation, hyperthermia

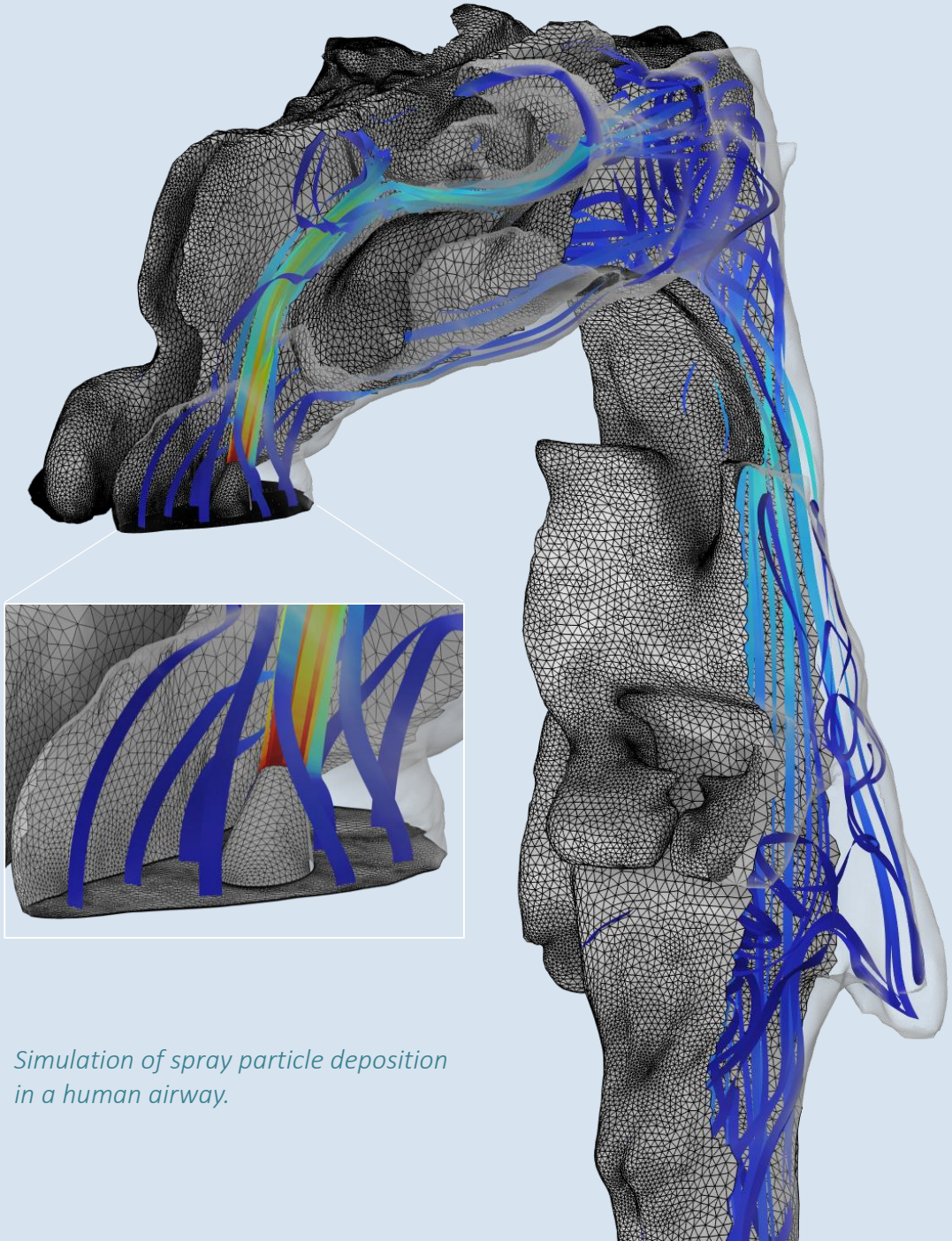
Sources and Formats of Mesh-Based Data

- Medical imaging
  - CT, MRI, ultrasound
  - Segmentation: MATLAB Medical Imaging Toolbox
- 3D scanning
  - Optical scanners
  - External anatomy, dental impressions
- Topology optimization
- STL, PLY and 3MF
  - Surface meshes from scanned data
  - Linear triangles



An STL Modeling Example

- Human airway surface mesh imported from an STL file
- Repair the mesh
  - Isolate and remove intersecting elements
  - Fill in the holes
- Intersect the mesh with planes to create inflow and outflow boundaries
- Draw a nasal sprayer in COMSOL Multiphysics® and combine with the mesh
- Create a 3D domain inside the watertight surface mesh for setting up the physics
- Generate a calculation mesh in the domain before solving

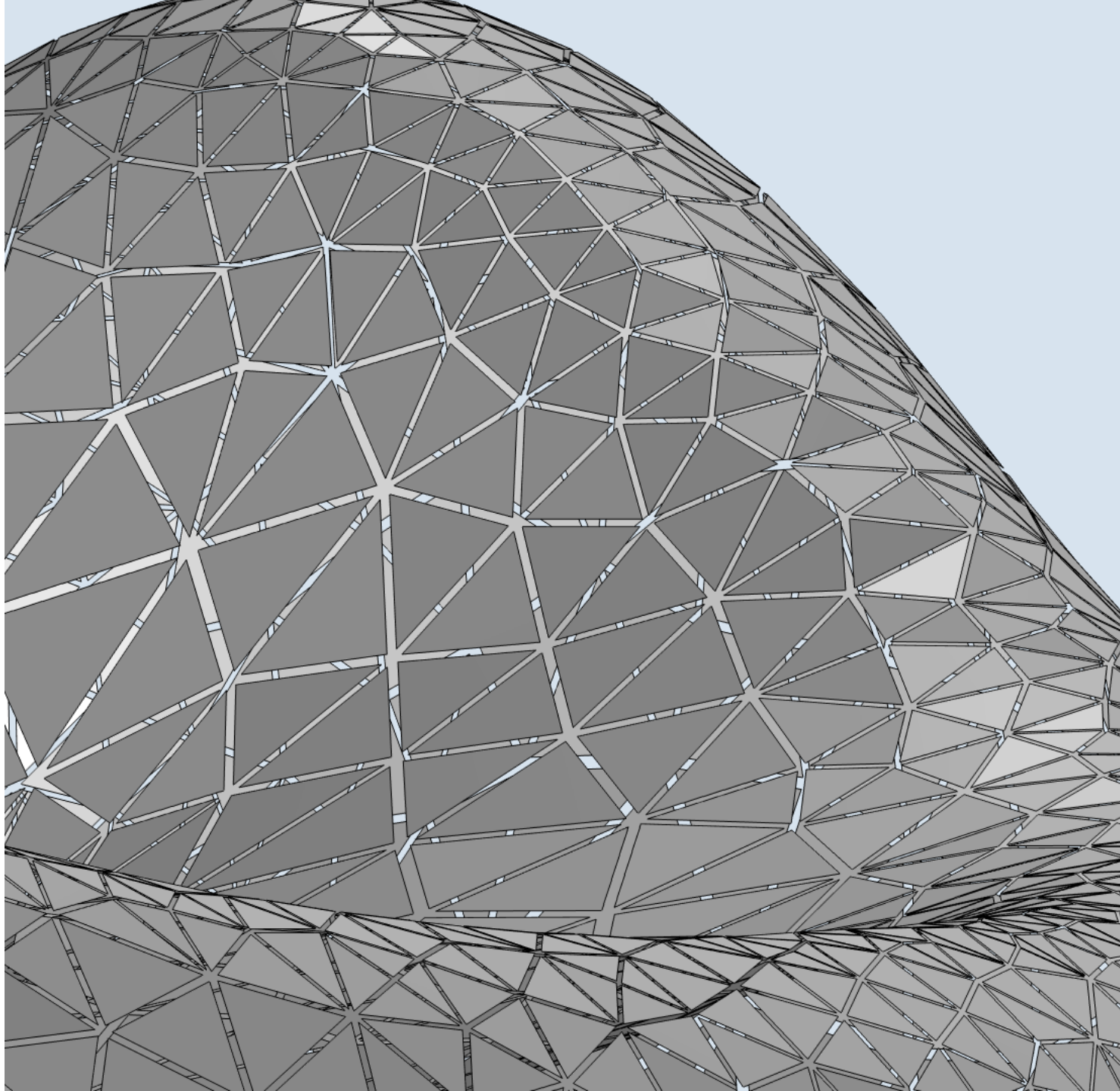


Simulation of spray particle deposition in a human airway.

The STL geometry of the airways is provided courtesy of Mark Yeoman, Continuum Blue, U.K.

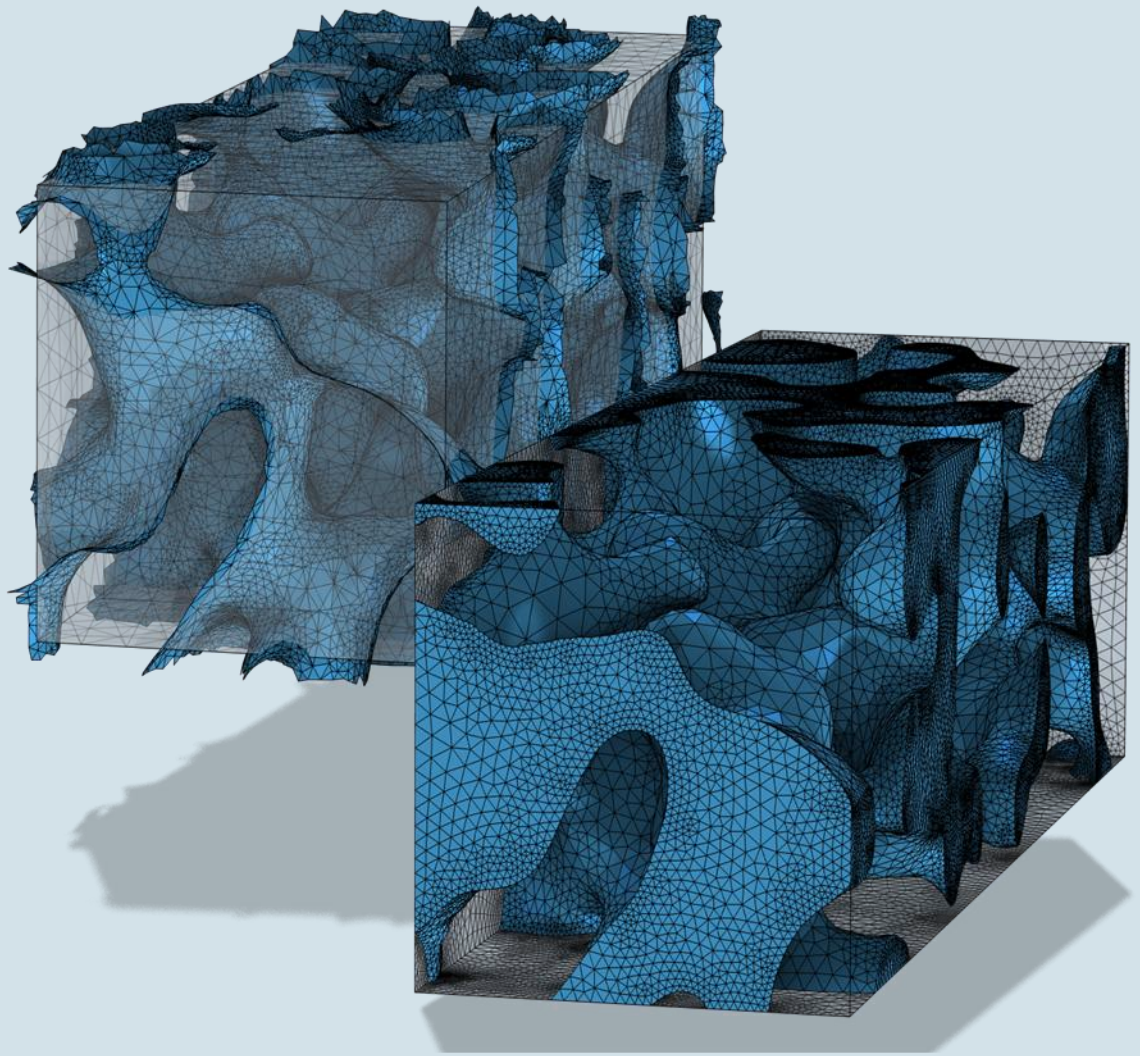
The STL Format

- Each mesh node specifies its own coordinate in file
  - Many duplicated coordinates
  - Tolerance used to stitch triangles together
- Domain and boundary information needs to be created during import or is created manually.
- The triangles in the file define a linear mesh, but the software will automatically curve the elements when needed.
- Imported STL meshes often need repair before simulation.



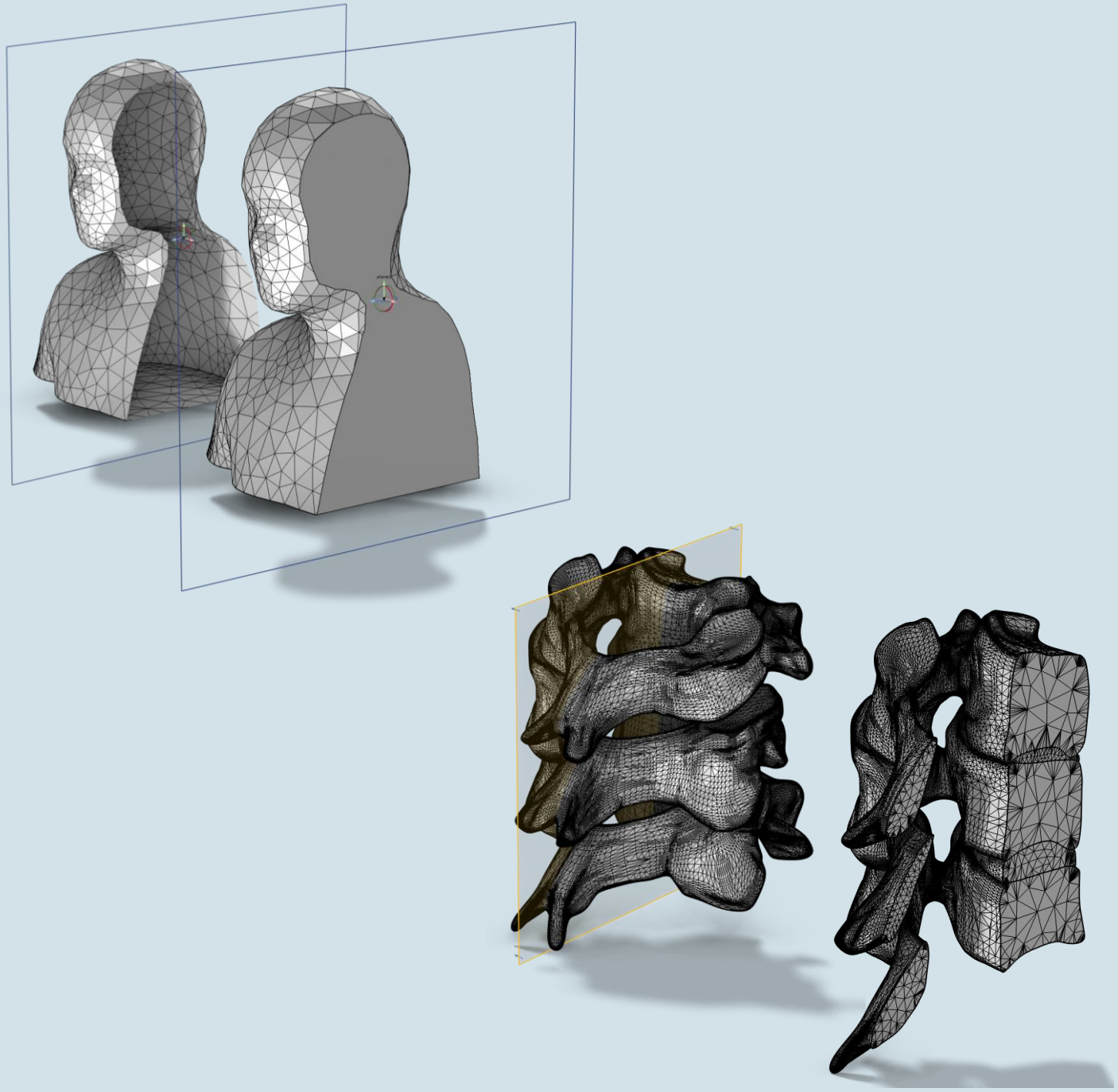
Editing and Repairing Meshes

- Repairing Meshes to Form 3D Domains
- Connecting Meshes
- Uniting and Partitioning
- Merging Meshes
- Remeshing Faces
- Creating Selections



Creating Domains from Surface Meshes

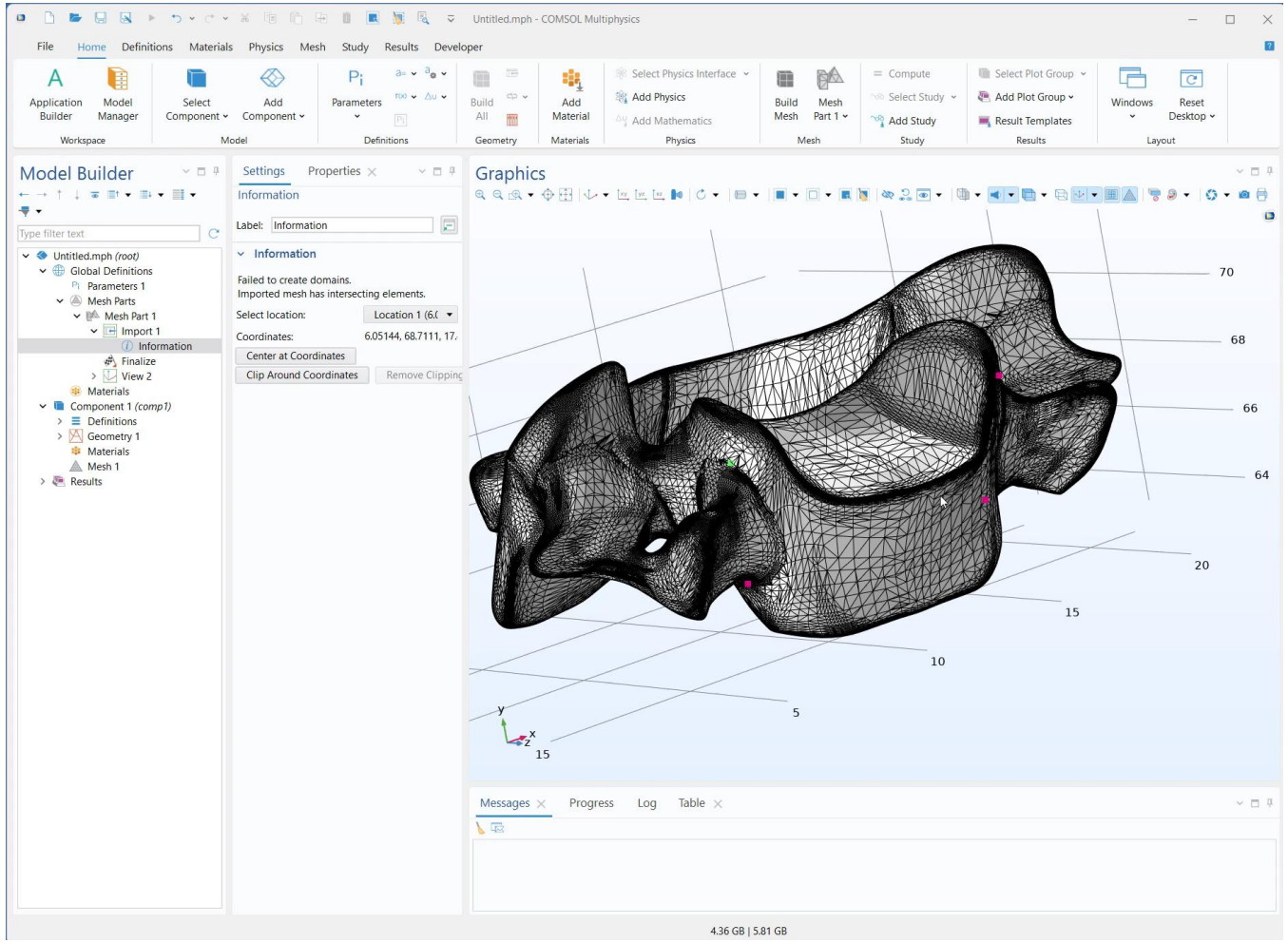
- Domains are volume regions where you can assign material properties and fill with a volume mesh.
- Several mesh editing operations automatically create domains within watertight surface meshes.
- Unite meshes to form new computational domains.



The STL geometry on the left is provided courtesy of Mark Yeoman, Continuum Blue, U.K.

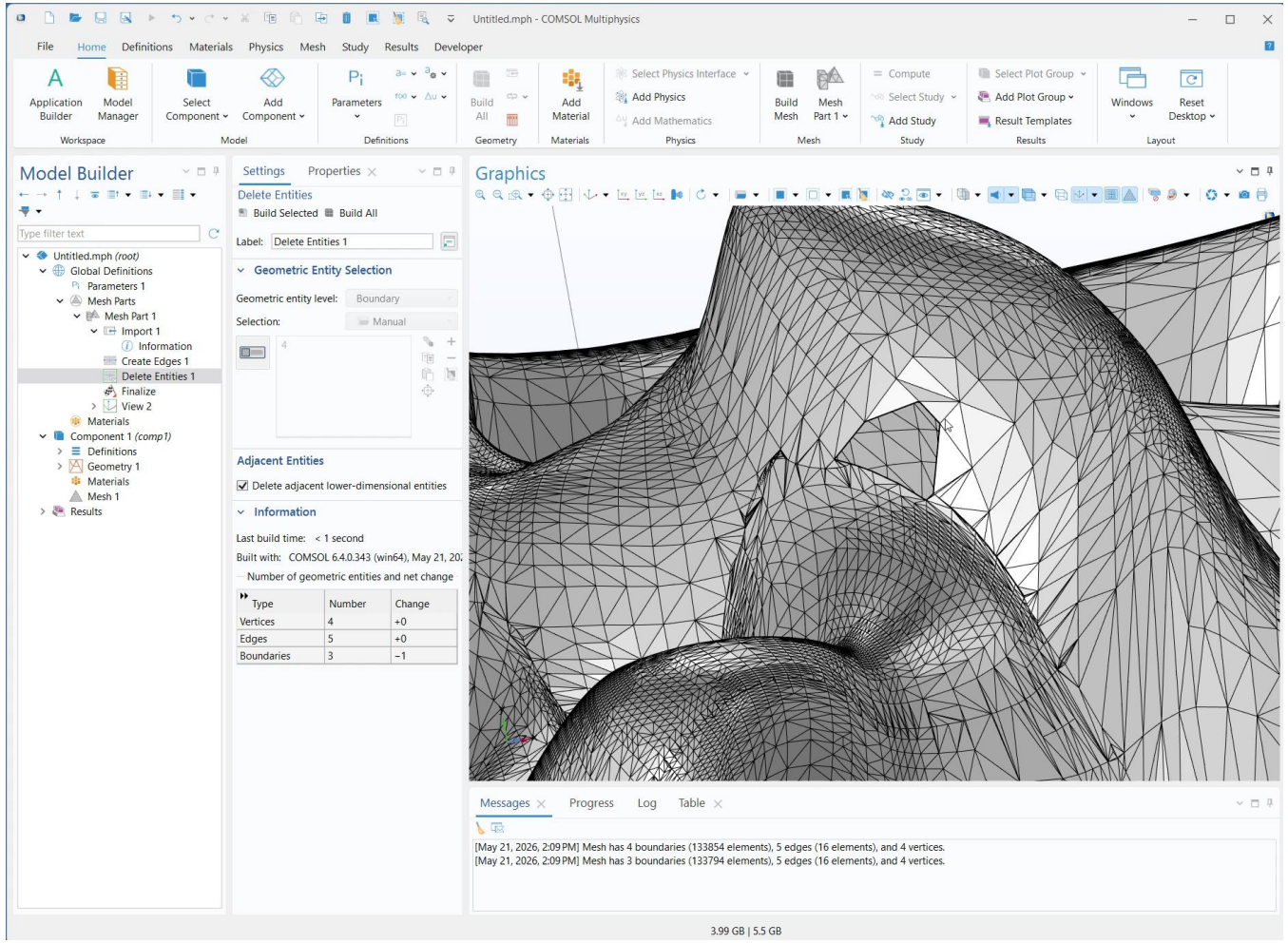
Identifying and Repairing Self-Intersections

- Center at and clip around coordinates listed in the *Information*, *Warning*, and *Error* nodes to locate problematic elements. These can be isolated using the *Create Edges* operation and then deleted.



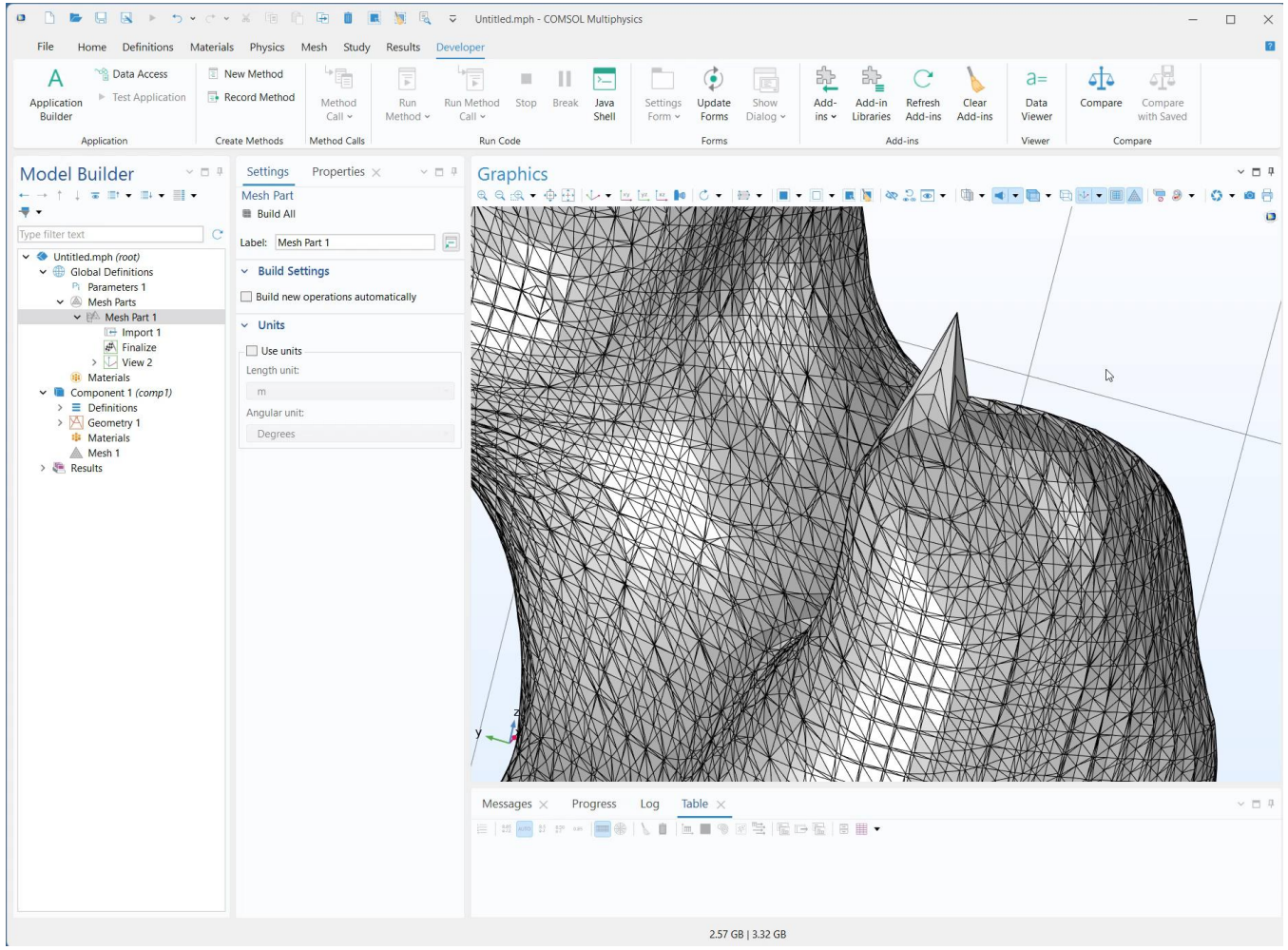
Repairing Holes

- Specify a tolerance for automatic repair of small holes during mesh import.
- Use the *Fill Holes* operation to automatically repair holes based on hole perimeters.
- Use the *Create Faces* operation to generate meshed faces after manually selecting the edges bounding the hole.



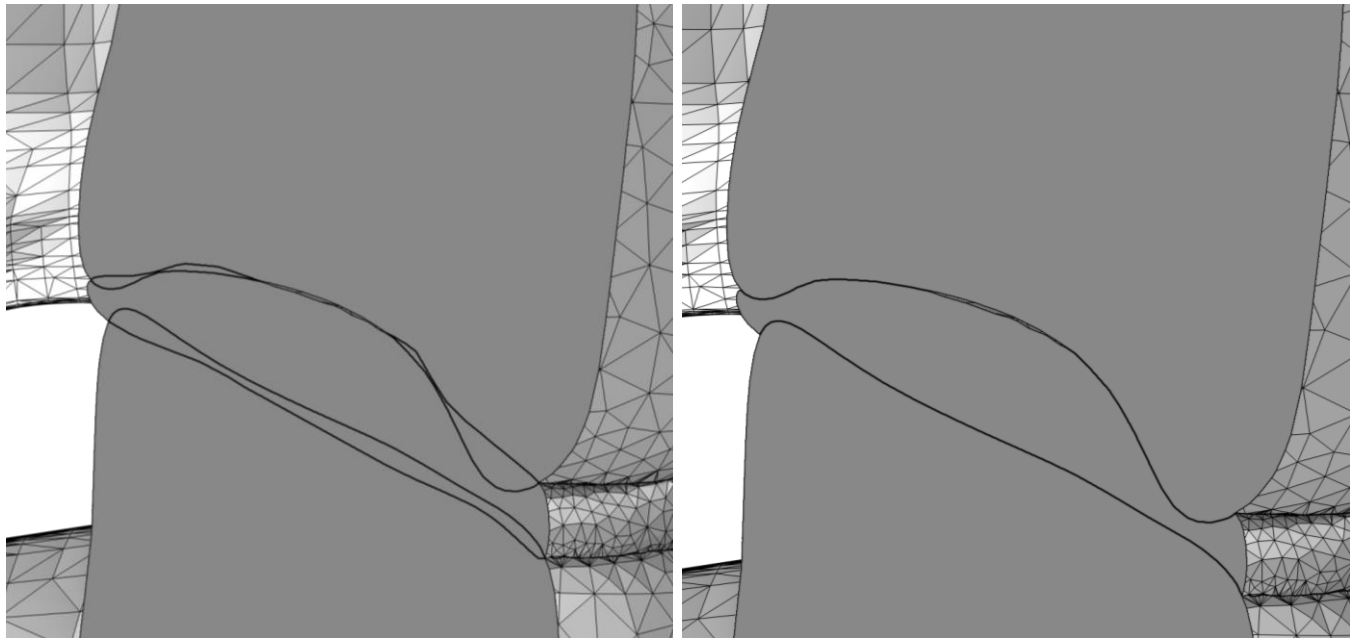
Removing Sharp Irregularities

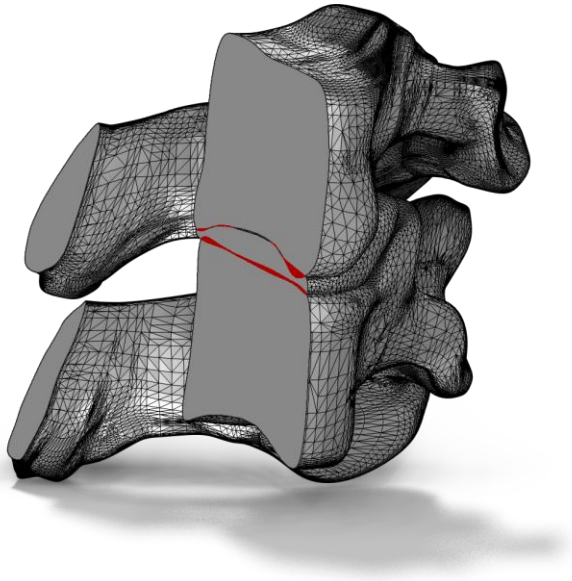
- The *Remesh Faces* operation provides automatic removal of smaller irregularities when remeshing faces.
- Manually isolate irregularities using the *Create Edges* operation, delete the surface, and replace it using the *Fill Holes* or the *Create Faces* operations.



Uniting Meshes

- Visualize overlapping domains using a *Clip Plane* with the *Show Cross Section* setting.
- Use the *Union* operation to combine meshes and calculate the intersections.
- Overlapping regions will form their own domains.
- The *Join Entities* operation is available to obtain appropriate domains for the simulation.

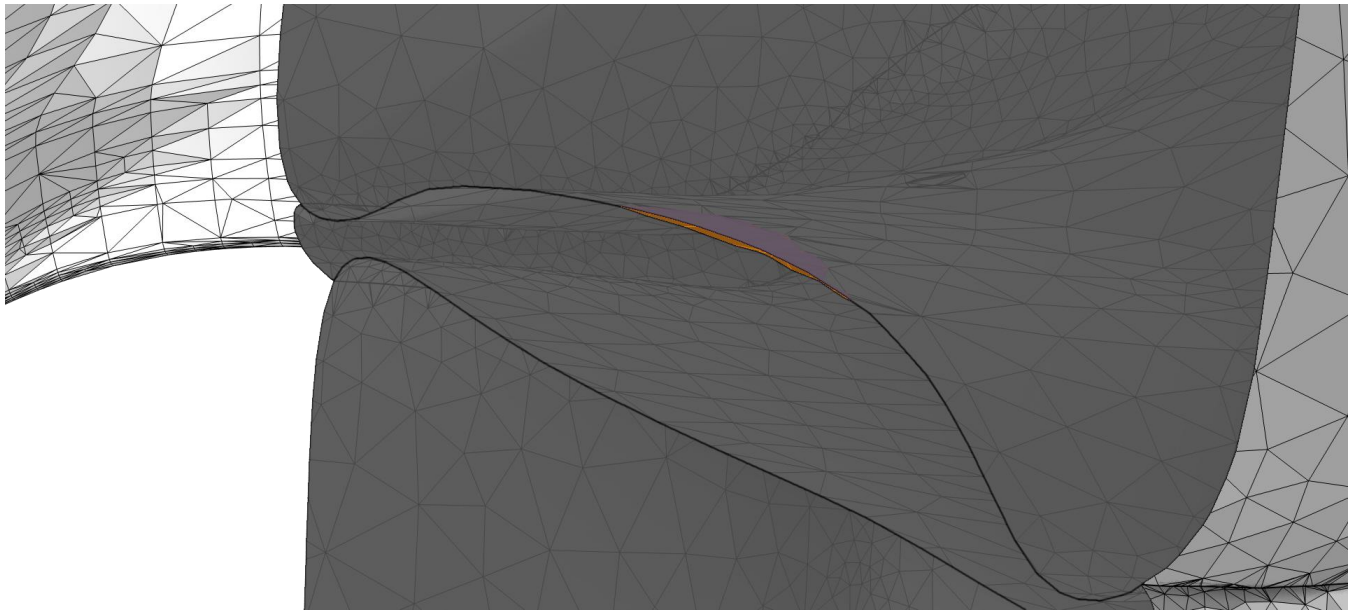




The STL geometry is provided courtesy of Mark Yeoman, Continuum Blue, U.K.

Merging Entities in Meshes

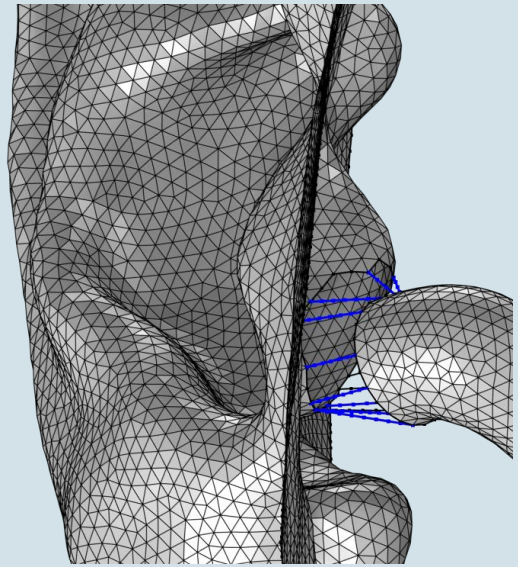
- Use the *Merge Entities* option to merge nearby faces, edges, or vertices in meshes, for example, for collapsing gaps or short edges.
- Optionally select which entities to keep when merging.

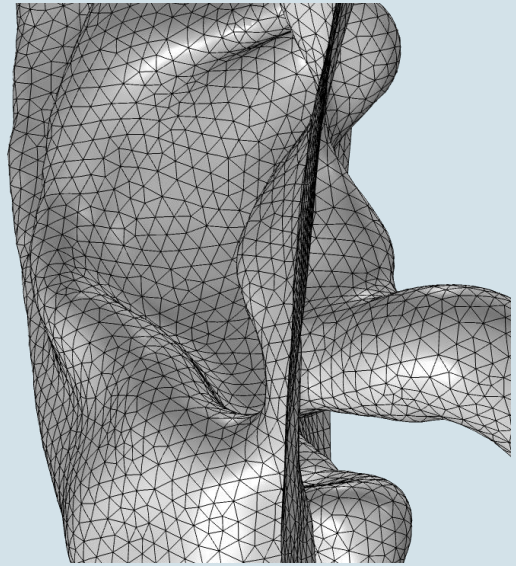


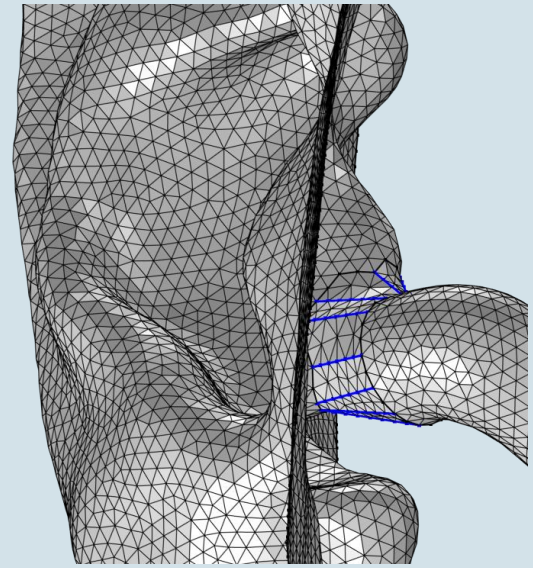
The STL geometry is provided courtesy of Mark Yeoman, Continuum Blue, U.K.

Connecting Meshes

- Bridge larger gaps between meshes by first creating meshed edges using the *Create Edges* operation, followed by the *Create Faces* operation. Next, you can use the *Join Entities* operation for an appropriate boundary partitioning and lastly, the *Remesh Faces* operation to smoothen the shape.

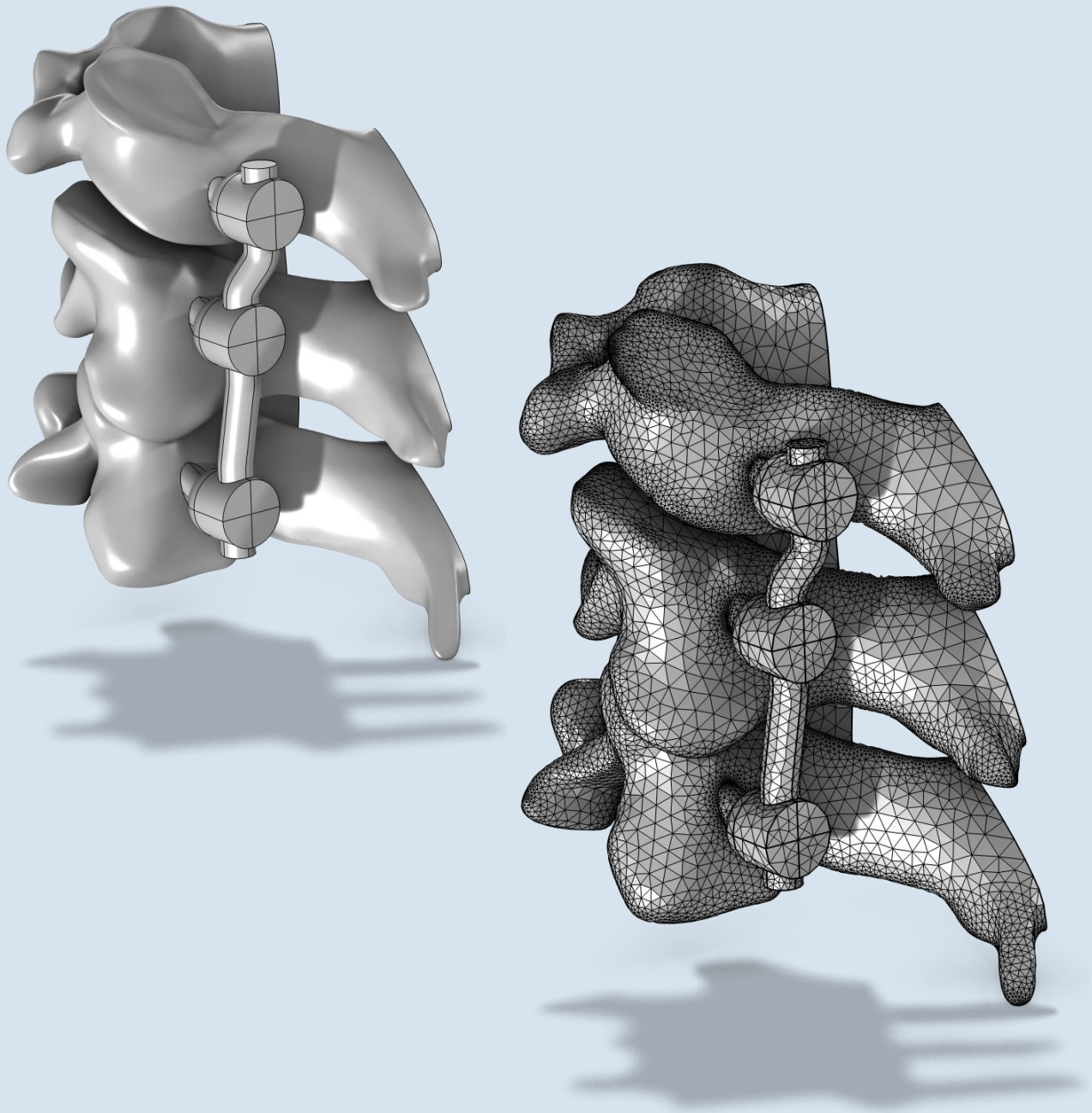






Showcase

- Import 3 vertebrae + 2 disc STL files
- Repair holes, misalignments, intersections
- Combine, intersect, and remesh
- Add pedicle screws and fixation rod



Untitled.mph - COMSOL Multiphysics

File Home Definitions Geometry Materials Physics Mesh Study Results Developer

Application Builder | Model Manager | Component 1 | Add Component | Parameters | Variables | Functions | Parameter Case | Variable Utilities | Equation Contributions | Build All | Import | LiveLink | Part Libraries | Add Material | Select Physics Interface | Add Physics | Add Mathematics | Build Mesh | Mesh 1 | Compute | Select Study | Add Study | Select Plot Group | Add Plot Group | Result Templates | Windows | Reset Desktop

Workspace | Model | Definitions | Geometry | Materials | Physics | Mesh | Study | Results | Layout

Model Builder

Type filter text

- Untitled.mph *(root)*
  - Global Definitions
    - Parameters 1
    - Materials
  - Component 1 *(comp1)*
    - Definitions
    - Geometry 1
    - Materials
    - Mesh 1
  - Results

Settings | Properties

Geometry

Build All

Label: Geometry 1

Units

Scale values when changing units

Length unit: m

Angular unit: Degrees

Reduction for Symmetry Boundaries

yz-plane: remove x<0

zx-plane: remove y<0

xy-plane: remove z<0

Cleanup

Automatic detection of small details

Advanced

Geometry representation: CAD kernel

Design Module Boolean operations

Default repair tolerance: Automatic

Build new operations automatically

Build automatically when leaving geometry

Graphics

Messages | Progress | Log | Table

6.14 GB | 8.62 GB

Untitled.mph - COMSOL Multiphysics

File Home Definitions Materials Physics Mesh Study Results Developer

Application Builder | Model Manager | Select Component | Add Component | Parameters | Variables | Functions | Parameter Case | Variable Utilities | Equation Contributions | Build All | Import | LiveLink | Part Libraries | Add Material | Select Physics Interface | Add Physics | Add Mathematics | Build Mesh | C5 | Compute | Select Study | Add Study | Select Plot Group | Add Plot Group | Result Templates | Windows | Reset Desktop

Workspace | Model | Definitions | Geometry | Materials | Physics | Mesh | Study | Results | Layout

Model Builder

Type filter text

- Untitled.mph *(root)*
  - Global Definitions
    - Parameters 1
    - Mesh Parts
      - C3
      - C3-C4 Disc
      - C4
      - C4-C5 Disc
      - C5
    - Materials
  - Component 1 *(comp1)*
  - Results

Settings | Properties

Untitled.mph

Protection

Editing not protected — Set Password

Running not protected — Set Password

Used Products

CAD Import Module

COMSOL Multiphysics

Unit System

SI

Presentation

Title:

Description:

Author:

Computation time

Expected:

Last:

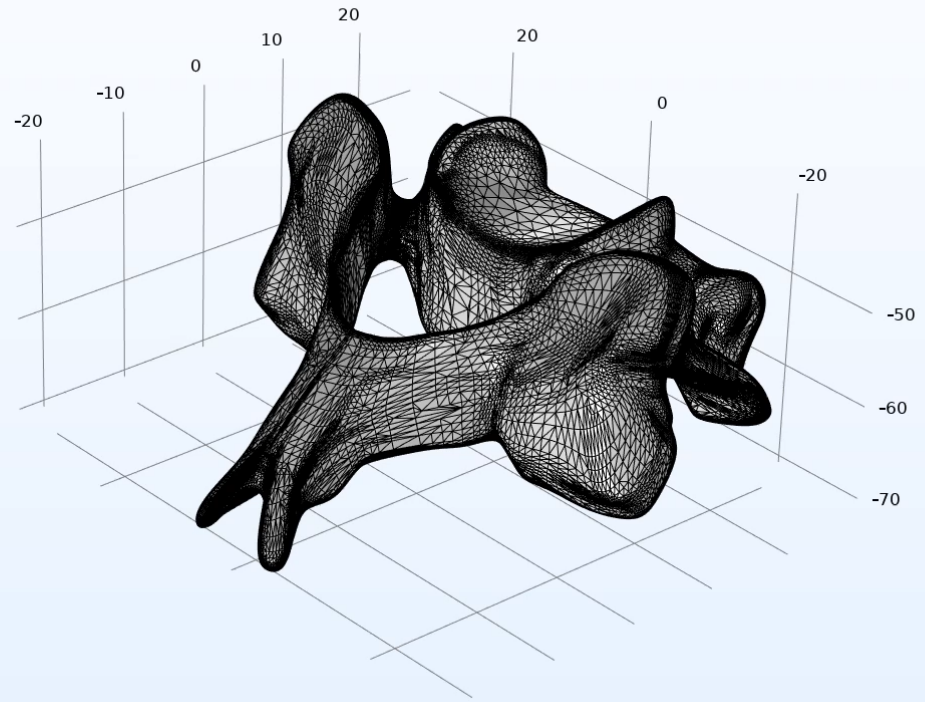
Thumbnail

Set from Graphics Window | Load from File...

Save

Built, computed, and plotted data

Graphics



Messages | Progress | Log | Table

[May 21, 2026, 4:22 PM] Imported mesh with 6312 boundary elements from sequence 'C4-C5 Disc'.
[May 21, 2026, 4:22 PM] Mesh has 4 domains, 6 boundaries (259598 elements), 8 edges (471 elements), and 8 vertices.
[May 21, 2026, 4:22 PM] Imported mesh with 146718 boundary elements from sequence 'C5'.
[May 21, 2026, 4:22 PM] Mesh has 5 domains, 7 boundaries (406316 elements), 8 edges (471 elements), and 8 vertices.
[May 21, 2026, 4:24 PM] Opened file: C:\Users\vrbata\Downloads\CM_2026\Untitled.mph

6.84 GB | 9.04 GB

Untitled1.mph - COMSOL Multiphysics

File Home Definitions Materials Physics Mesh Study Results Developer

Application Builder | Model Manager | Select Component | Add Component | Parameters | Variables | Functions | Parameter Case | Variable Utilities | Equation Contributions | Build All | Import | LiveLink | Part Libraries | Add Material | Select Physics Interface | Add Physics | Add Mathematics | Build Mesh | Combined Mesh | Compute | Select Study | Add Study | Select Plot Group | Add Plot Group | Result Templates | Windows | Reset Desktop

Workspace | Model | Definitions | Geometry | Materials | Physics | Mesh | Study | Results | Layout

Model Builder

Type filter text

- Untitled1.mph *(root)*
  - Global Definitions
    - Parameters 1
    - Mesh Parts
      - C3
      - C3-C4 Disc
      - C4
      - C4-C5 Disc
      - C5
      - Combined Mesh
    - Materials
  - Component 1 *(comp1)*
  - Results

Settings | Properties

Untitled1.mph

Protection

Editing not protected — Set Password

Running not protected — Set Password

Used Products

CAD Import Module

COMSOL Multiphysics

Unit System

SI

Presentation

Title:

Description:

Author:

Computation time

Expected:

Last:

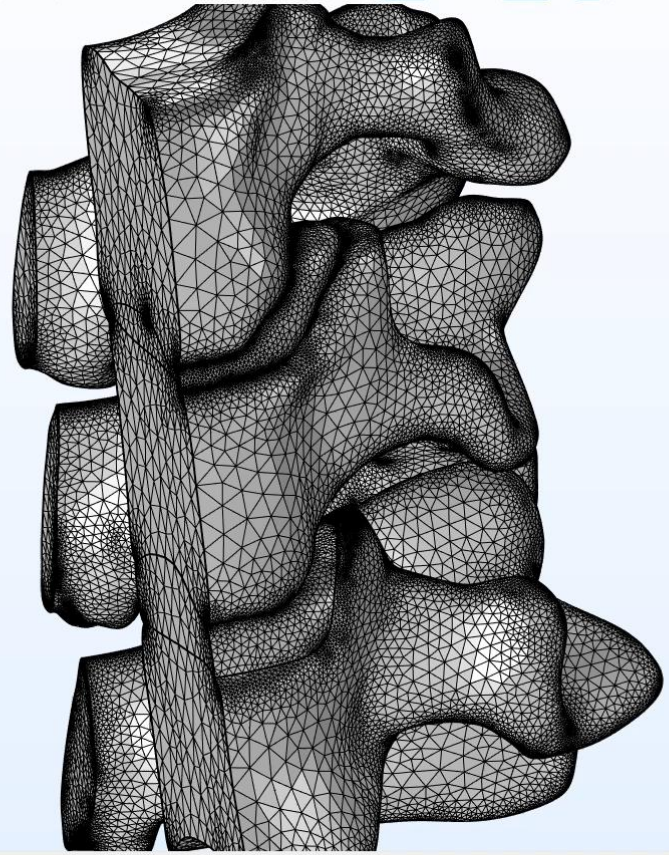
Thumbnail

Set from Graphics Window | Load from File...

Save

Built, computed, and plotted data

Graphics



Messages | Progress | Log | Table

[May 21, 2026, 4:30 PM] Mesh has 10 domains, 26 boundaries (406001 elements), 33 edges (2715 elements), and 21 vertices.
[May 21, 2026, 4:31 PM] Mesh has 5 domains, 17 boundaries (203818 elements), 26 edges (1702 elements), and 18 vertices.
[May 21, 2026, 4:32 PM] Mesh has 5 domains, 17 boundaries (148082 elements), 26 edges (1629 elements), and 18 vertices.
[May 21, 2026, 4:33 PM] Saved file: C:\Users\vrbata\Downloads\CM_2026\Untitled1.mph (48 MB)

7.42 GB | 10.13 GB

Try It Yourself

- Step-by-step PDF guide
- 5 STL files: 3 vertebrae + 2 discs
- Build the model from scratch
- Link: https://www2.humusoft.cz/KCM2026/meshing/

- Questions?