LA-UR-23-25480

Approved for public release; distribution is unlimited.

Title:	CrossLink: Meshing a 3D part from a STEP file
Author(s):	Long, Louie Joseph Drayna, Travis William Lang, Laura M. Stam, Henry Russell
Intended for:	Presentation for tutorial purposes
Issued:	2023-05-19

CrossLink

Meshing a 3D part from a STEP file

Travis Drayna
Laura Lang
Louie Long
Henry Stam

March 13, 2023

National Nuclear Security Administration
Managed by Triad National Security, LLC for the U.S. Department of Energy's NNSA

Overview

- This example shows how to use CrossLink to create a mesh for a 3D geometry part read in from a STEP file
- A STEP file (Standard for the Exchange of Product data) is a common file format used for 3D modelling that can be written out from a CAD program (e.g., Creo Parametric)
- Being able to read and mesh STEP file geometries is essential for meshing parts from engineering models
- Creating the mesh typically involves:
  - Importing the geometry
  - Creating geometry groups
  - Assigning curves and surfaces to the geometry groups
  - Building a topology
  - Applying geometric and mesh constraints
  - Generating the final mesh
- Current issues with this process in CrossLink include:
  - The GUI does not display trimmed surfaces
  - The user must manually create and assign geometry groups
  - The user must be aware of duplicate curves and surfaces from the CAD model

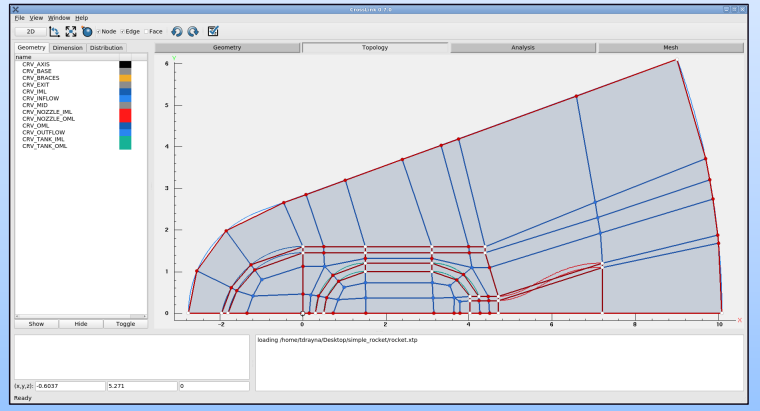
CrossLink – Scalable 2D/3D Topology-based Meshing

Geometry

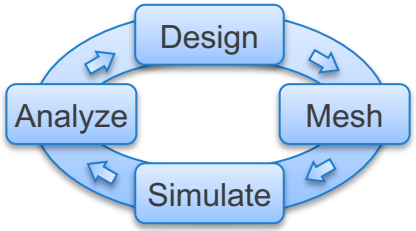
xGeom
- Python API
- NURBS model
- 2D/3D geometry
- Flexible I/O

xCAD
- Python API
- Native CAD driver
- Creo Parametric
- C++ toolkit

CrossLink – GUI



xMesh – Mesh Engine
- Topology-based mesh engine
- Parallel mesh optimization



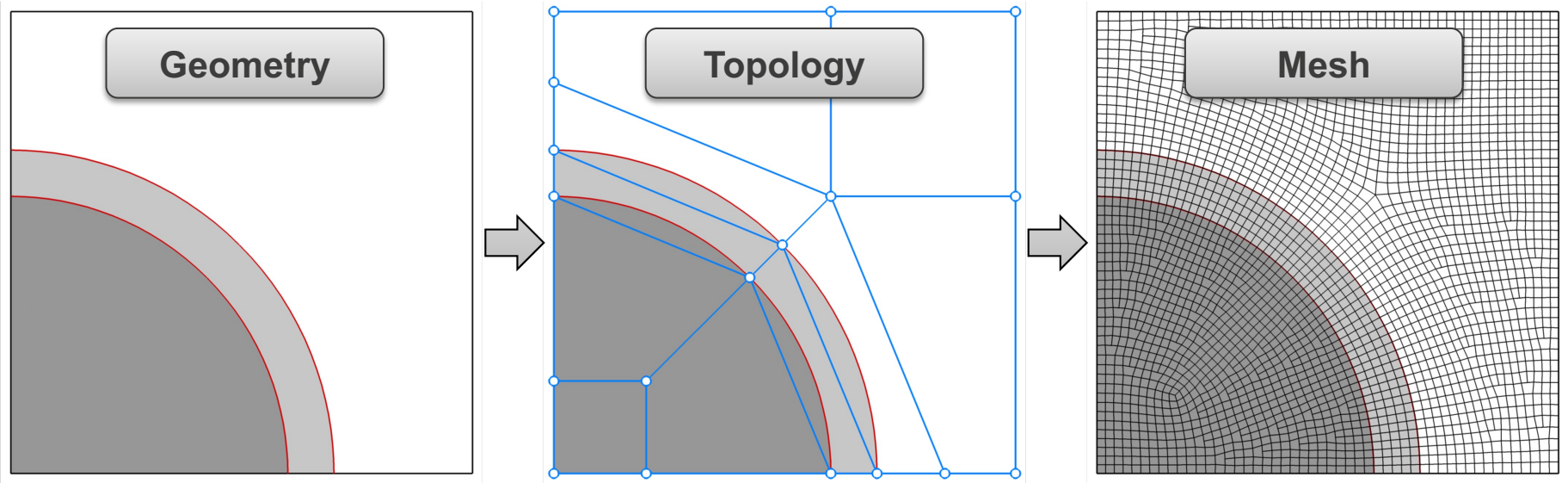
Multi-physics Code

Mesh File

Mesh Optimizer
- Improved simulation robustness
- Improved mesh quality control

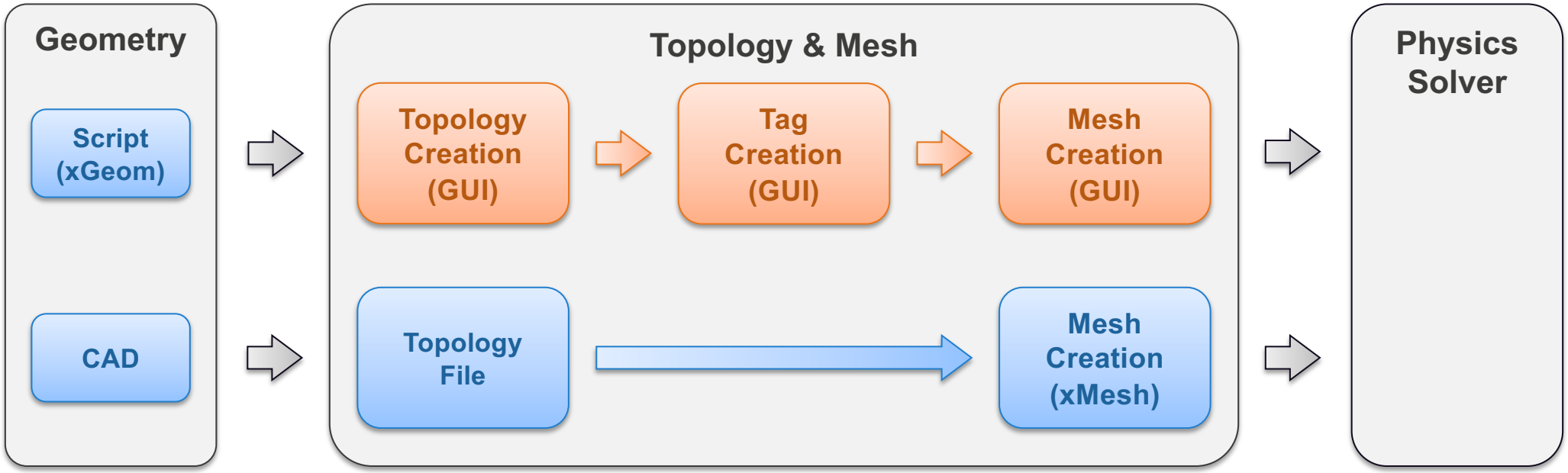
ParMesh API
- C++ library
- Reconstruct mesh in memory
- Scalable mesh size
- Simplified startup and partitioning
- Supports linear and curved elements
- Allows for structured memory layout

What is the meshing process?



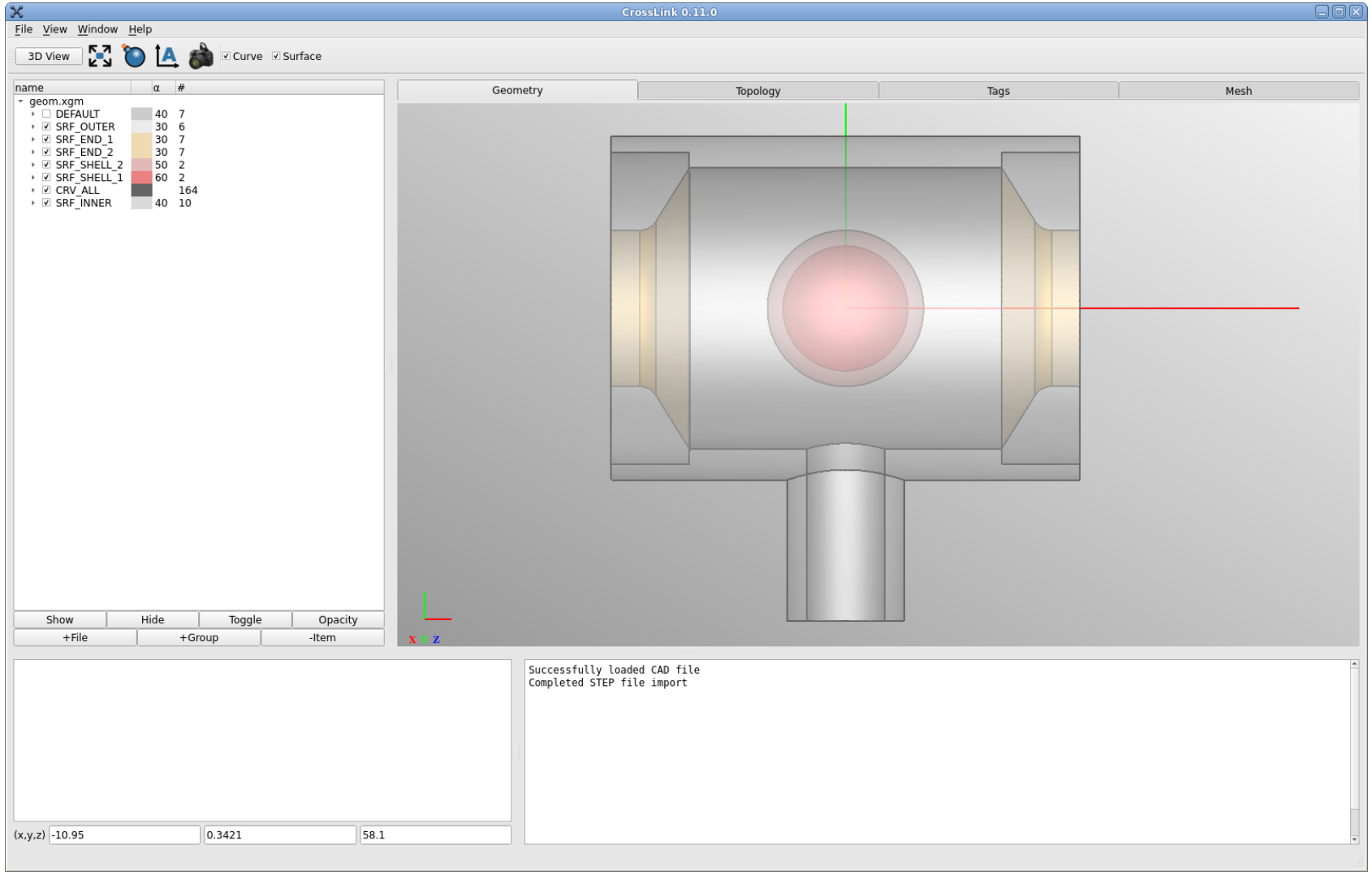
1. Starting with geometry, geometry groups are used to define hard internal and external boundaries.
2. A mesh topology (nodes, edges, faces, and blocks) is created to decompose the spatial domain into regular entities.
3. Geometric constraints are imposed by assigning topological entities to geometry groups.
4. Mesh rules (e.g., dimension, distribution, spacing, feathering) are defined to control mesh quality.
5. Mesh tags (e.g., boundary, volume) are defined for the desired simulation (e.g., CFD, FEA).
6. The mesh is initialized, optimized, and exported to a mesh file.

General Workflow



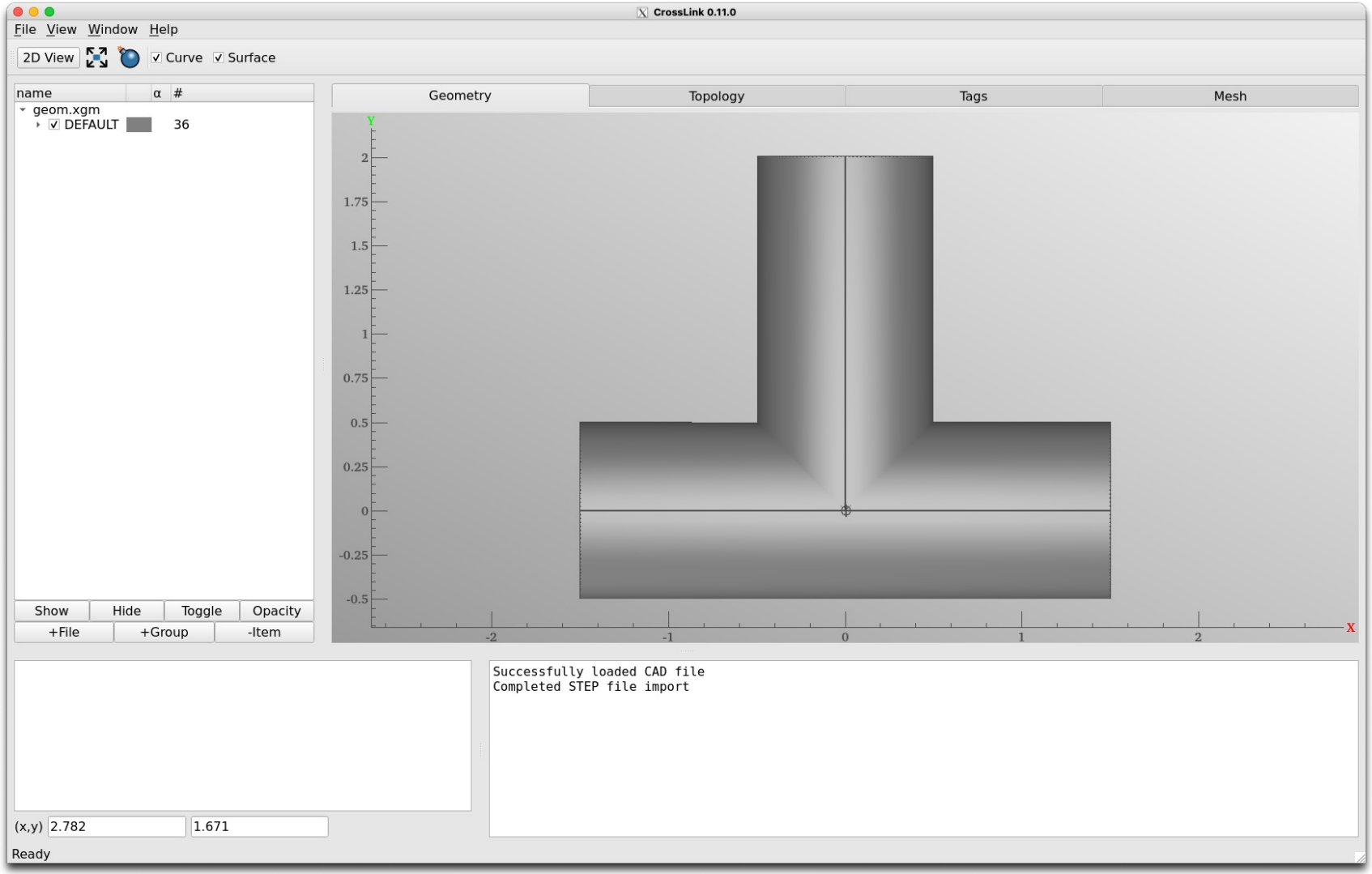
- Initial geometry is created via a geometry script or a CAD software package.
- A mesh topology is created interactively using CrossLink's Graphical User Interface (GUI).
- Geometry constraints, meshing rules, and mesh tags are defined and stored in the topology file.
- The xGeom and xMesh python API libraries are used to automate the mesh generation process.
- The workflow handles geometric parameter changes and robustly (re)meshes each design.

CAD Geometry



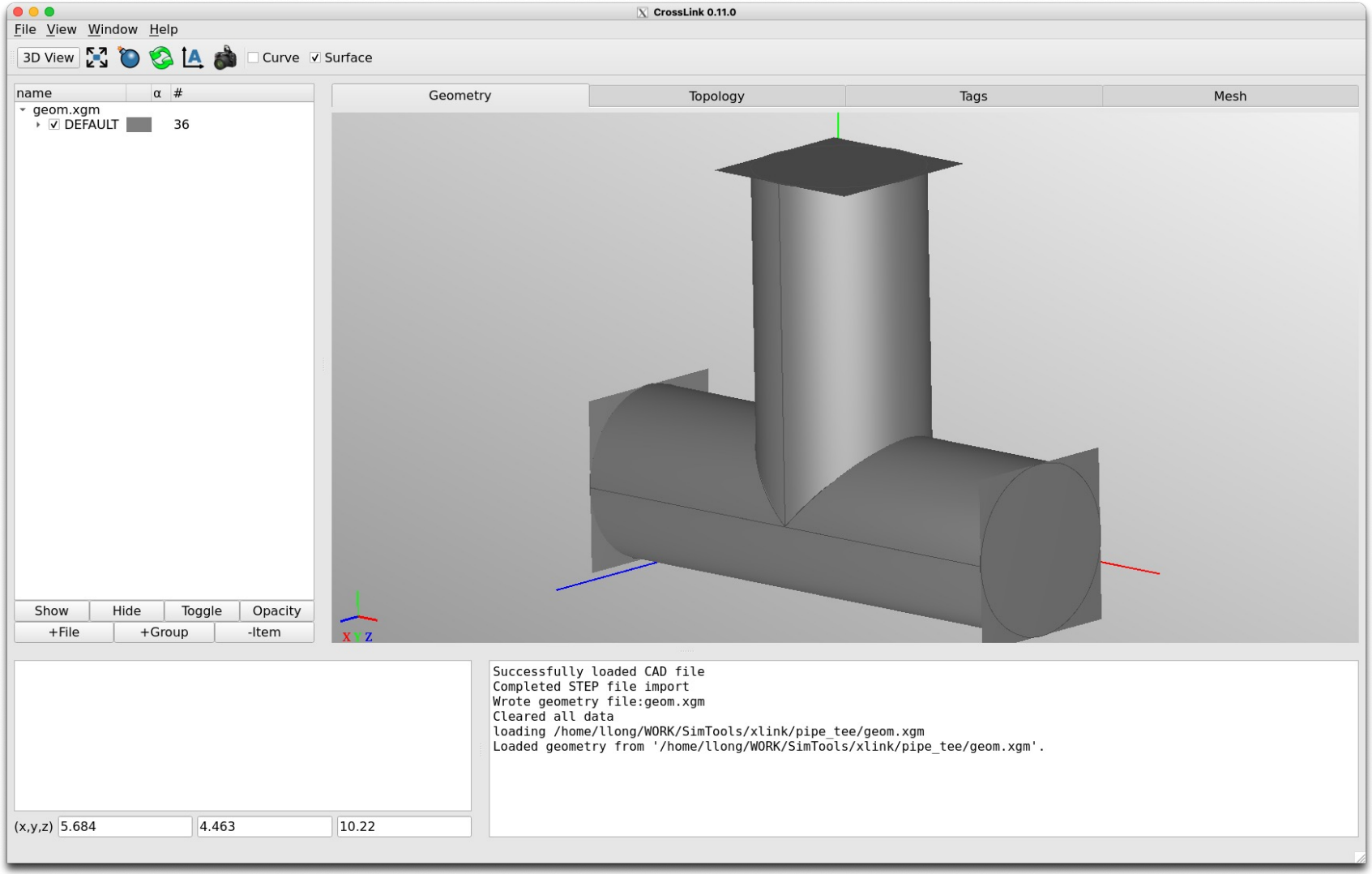
- CrossLink handles geometry created with CAD software.
- These models are translated as BREP NURBS models.
- Supported data exchange format is STEP.

Step 1: Import the STEP file



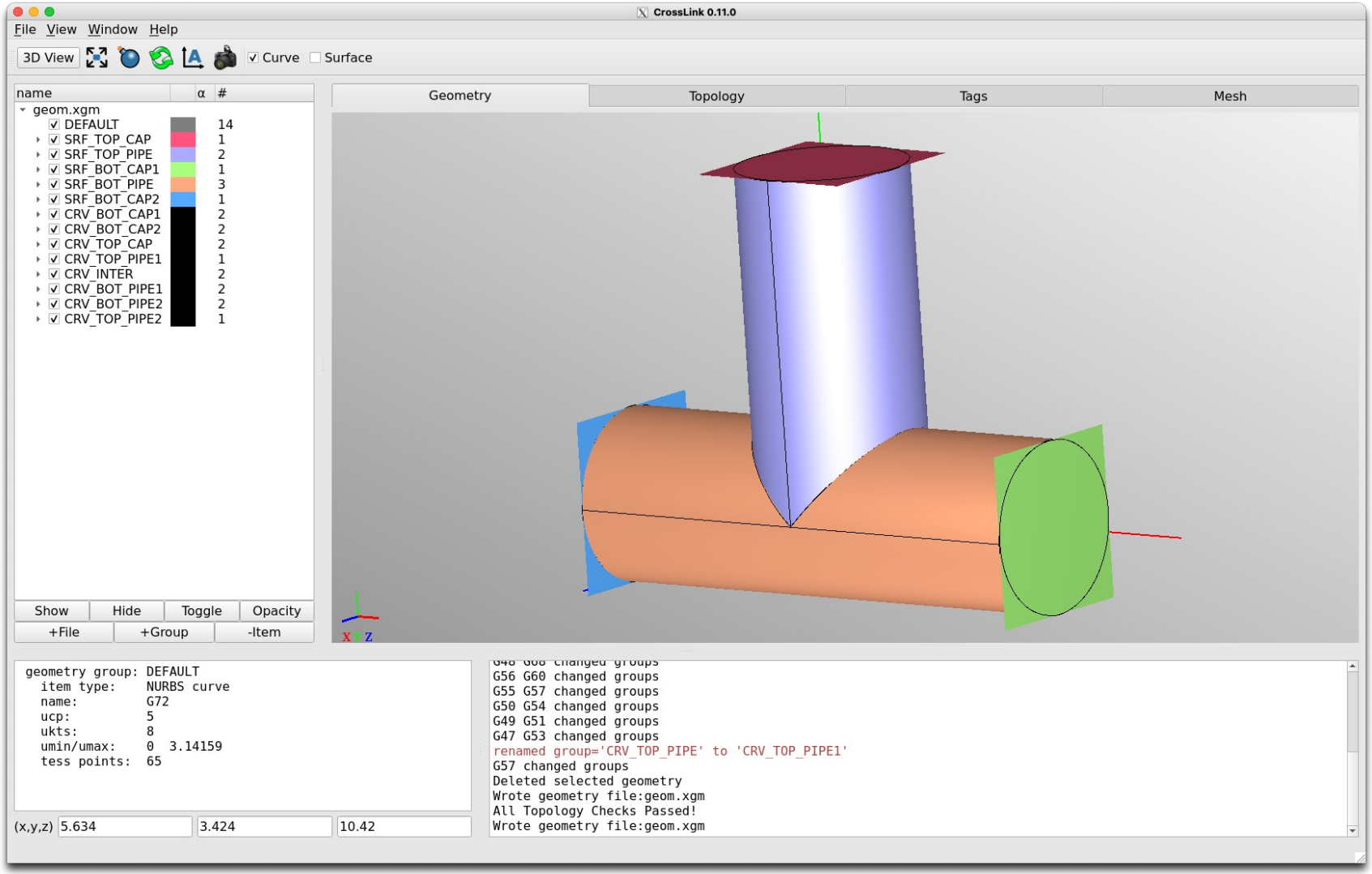
- Open the CrossLink GUI
- Import the STEP file `(ctrl + i)`

Step 2: Save the geometry file



- Save the geometry file
- Clear your workspace
- Reload
- Note that the surfaces are now untrimmed. This is because we do not currently have the triangulation rendered in the GUI. The data is truly trimmed, but it is not rendered as such.

Step 3: Assign curve and surface groups

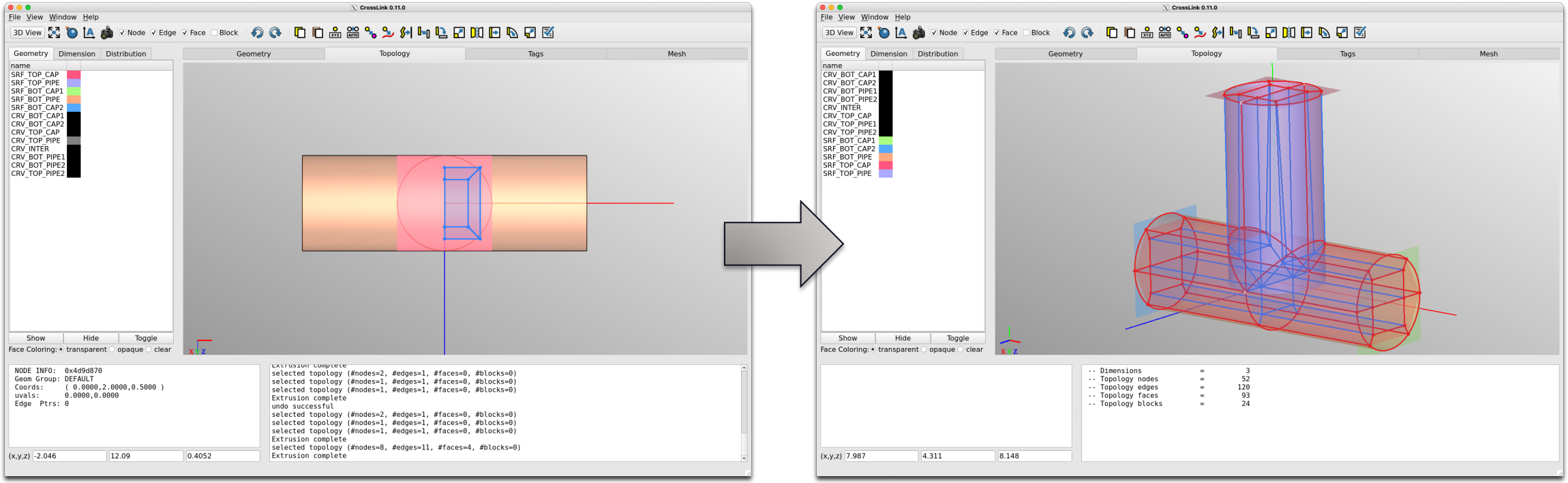


- In the geometry workbench, assign curve and surface groups and add entities to them. Note that the STEP file will contain duplicate curves. You can deal with these in one of two ways:

  1. Delete the duplicate curves from the DEFAULT group, or

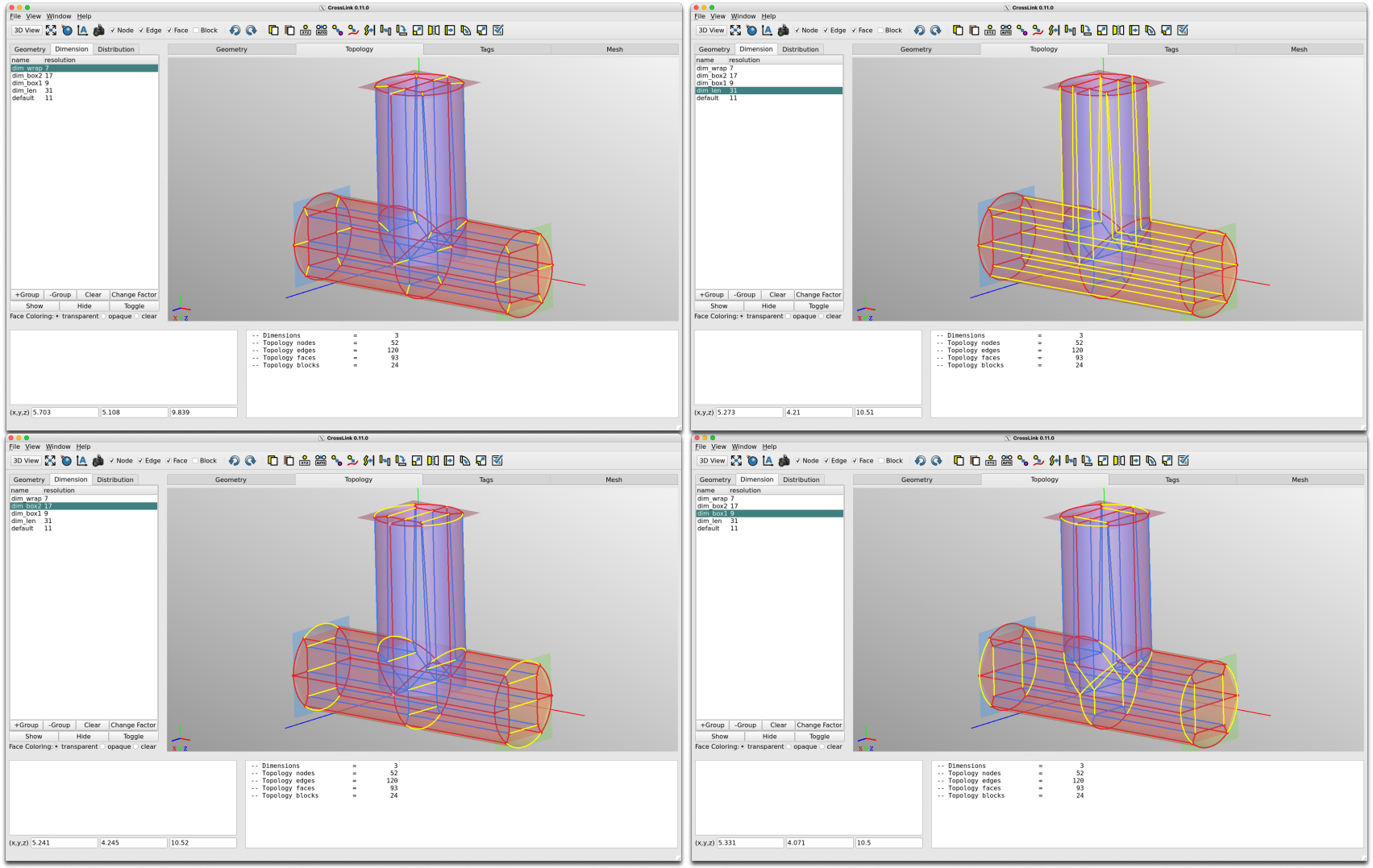
  2. Leave the duplicate curves in DEFAULT and hide the group

Step 4: Build the topology



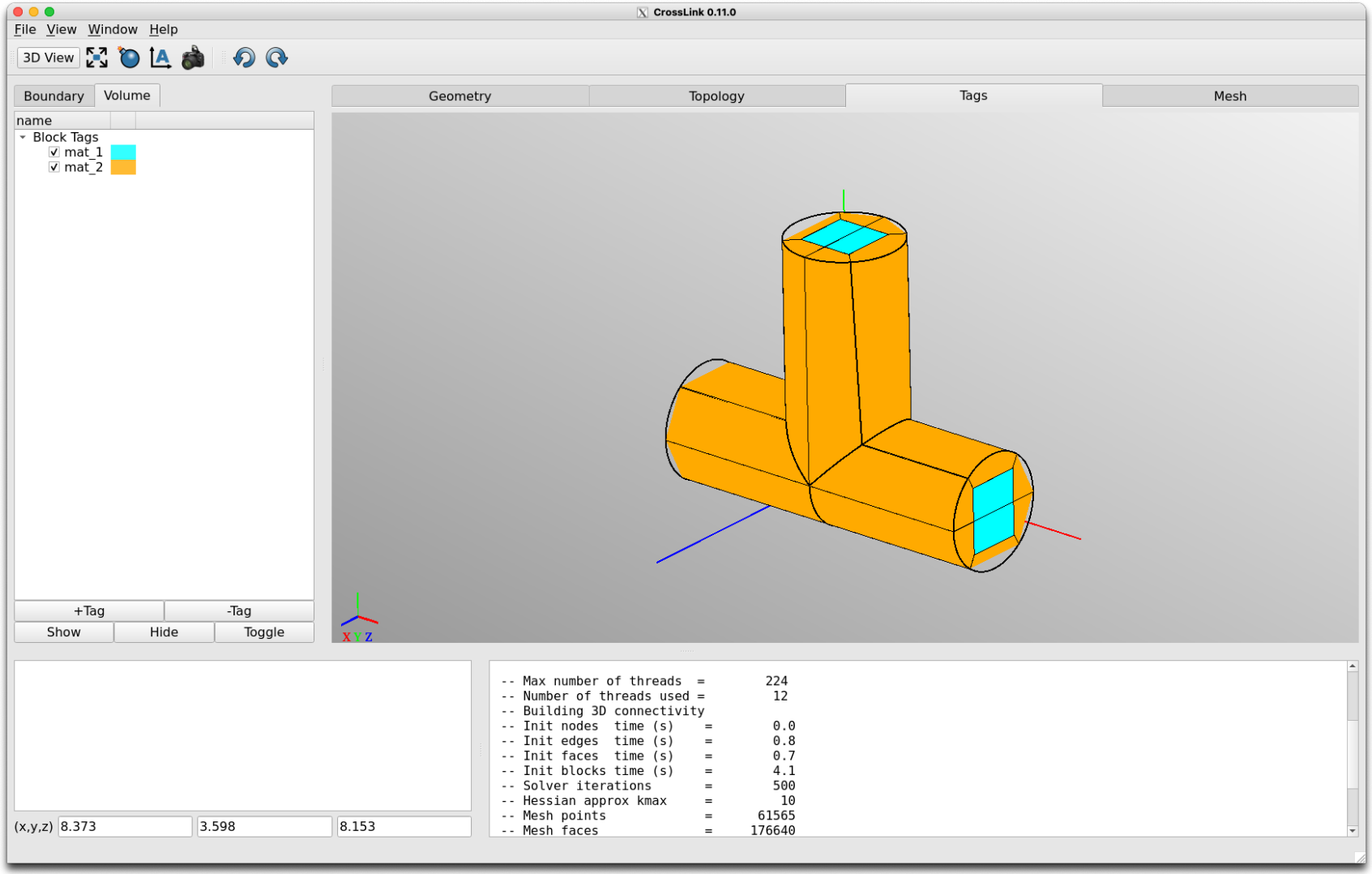
- Move to the topology workbench and start building the topology.
- The recommended approach is to start from the end of the perpendicular pipe and work your way inward.

Step 5: Set dimensions



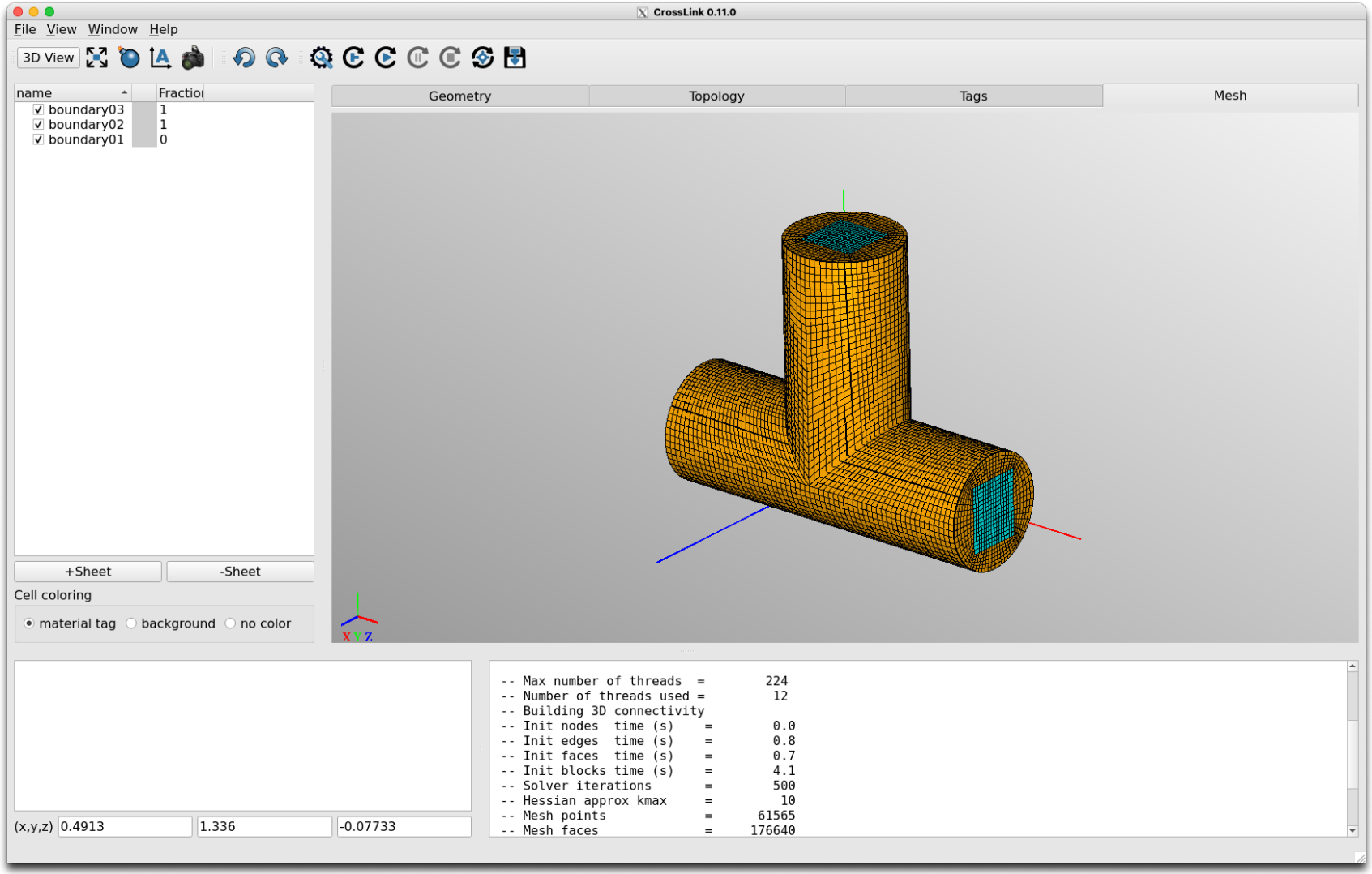
- In the topology workbench, assign dimension groups
- Right click on a curve and select “set dimension group”

Step 6: Set material tags



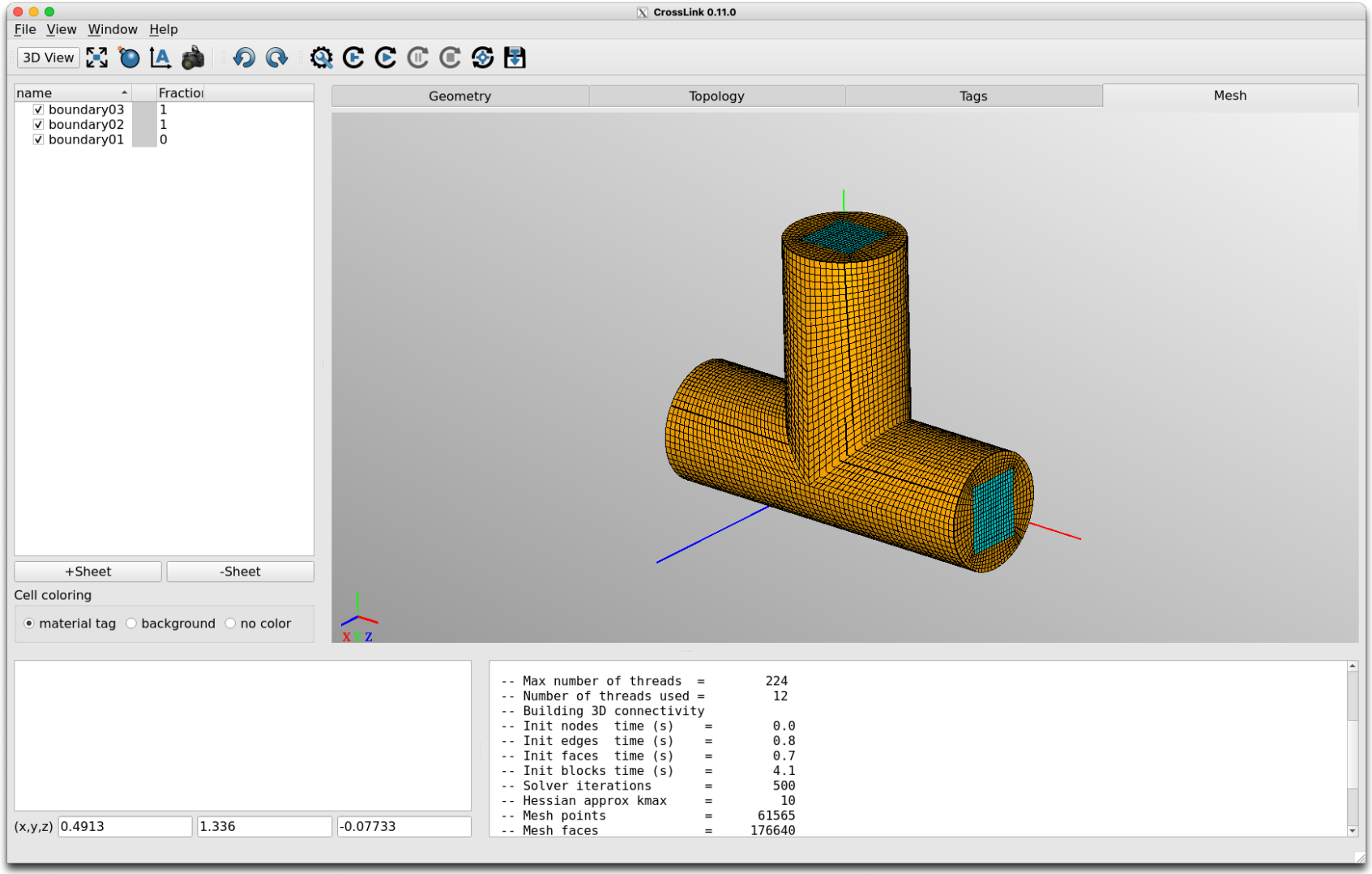
- Move to the tags workbench
- Select “volume” in the tree
- Set material tags

Step 7: Generate the mesh



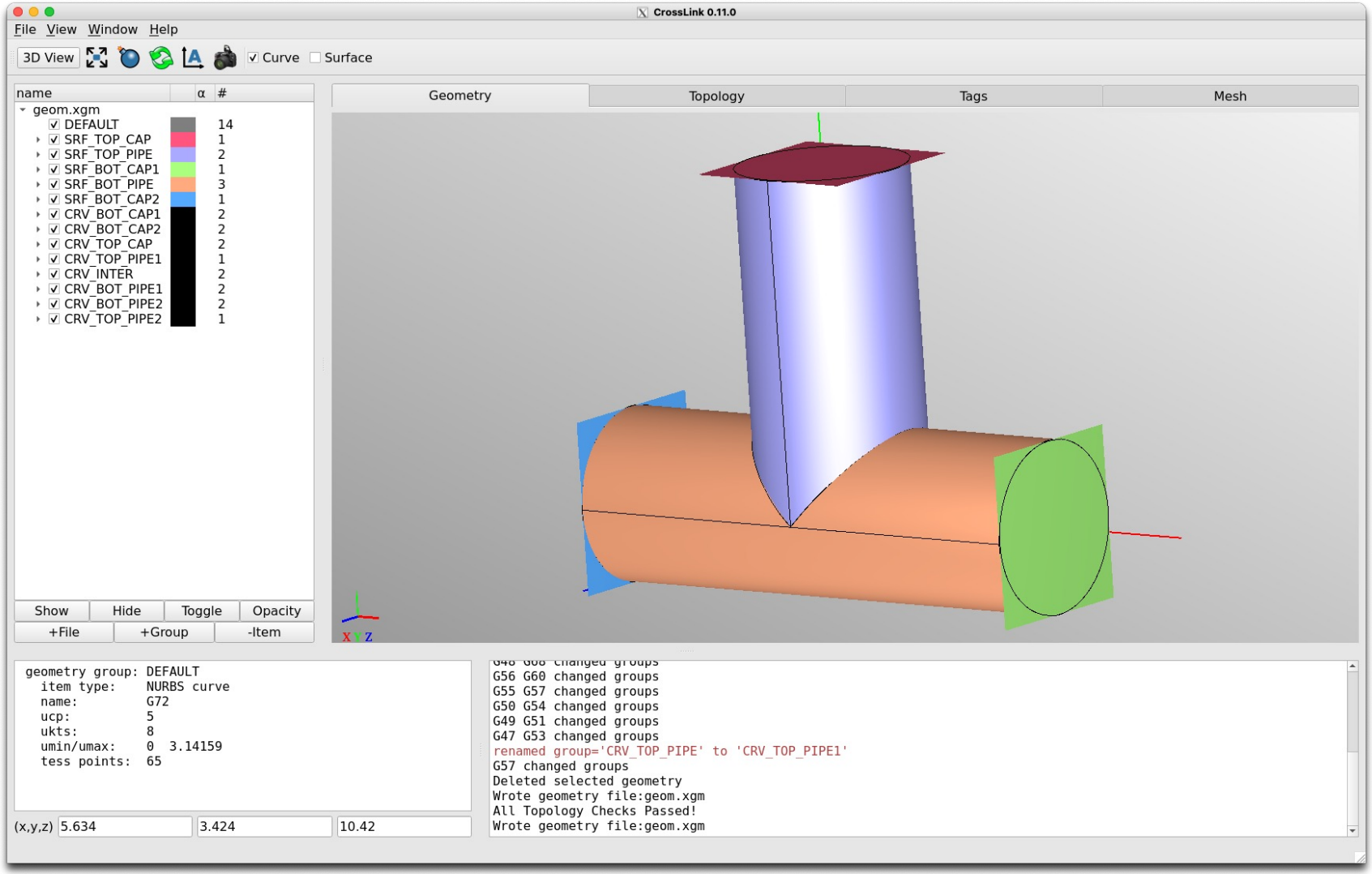
- Move to the mesh workbench
- Optionally, you can adjust the resolution and spacing as needed by jumping back and forth between the topology and mesh workbench.
- When you are satisfied with the mesh settings, run the optimizer to smooth the mesh.

Step 8: Export the mesh



- Click the ”export mesh” button to export the mesh
- Supported filetypes include X3D and Abaqus

Known issues with the STEP to mesh process



1. The GUI does not display trimmed surfaces
2. The user must manually create and assign geometry groups
3. The user must be aware of duplicate curves and surfaces from the CAD model